NASA Technical Memorandum 80182

SUPERSEDES NASA TM-80182, ISSUED JANUARY 1980

PASCO: STRUCTURAL PANEL ANALYSIS AND SIZING CODE,

USER'S MANUAL

Melvin S. Anderson, W. Jefferson Stroud,

Barbara J. Durling, and Katherine W. Hennessy

November 1981

CONTENTS

CONTENTS - Concluded

PASCO: STRUCTURAL PANEL ANALYSIS AND SIZING CODE , USER'S MANUAL

Melvin S. Anderson, W. Jefferson Stroud,
Barbara J. Durling, and Katherine W. Hennessy

SUMMARY

A computer code denoted PASCO is described. Buckling and vibration analyses are carried out with a linked-plate analysis computer code denoted VIPASA, which is included in PASCO. Sizing is based on nonlinear mathematical programming techniques and employs a computer code denoted CONMIN, also included in PASCO. Design requirements considered are initial buckling, material strength, stiffness, and vibration frequency. The report serves as a user's manual for PASCO.

INTRODUCTION

This report serves as a user's manual for a computer code denoted PASCO - Panel Analysis and Sizing COde. The code can be used to analyze and size prismatic structures having an arbitrary cross section. Its primary focus is stiffened panels made of laminated orthotropic materials. This tool is of particular value in analyzing and sizing filamentary composite structures.

The capability of PASCO and the approach used in the structural analysis and sizing are described in reference 1. When used in the analysis mode, PASCO calculates laminate stiffnesses, lamina stresses and strains (including the effect of temperature

and panel bending), buckling loads, vibration frequencies, and overall panel stiffness. When use: in the sizing mode, PASCO adjusts sizing variables to provide a low-mass panel design that carries a set of specified loadings without exceeding buckling or material strength allowables and that meets other design requirements such as upper and lower bounds on sizing variables, upper and lower bounds on overall bending, extensional and shear stiffnesses, ar.1 lower bounds on vibration frequencies.

The report begins with a discussion of structural modeling with PASCO. Plotting is then discussed briefly. Computer program definitions (including input and output) and organization are then presented. Finally, several illustrative examples are presented.

MODELING WITH PASCO

The modeling approach used in PASCO is basically the same as that used in VIPASA (refs. 2 to 4). Although emphasis is placed on flat panels having several identical bays the only restriction on configuration is that the structure is assumed to be prismatic (fig. 1). In addition, it is assumed that loads and temperatures do not vary along the length of the panel. Input is in NAMELIST format.

The general procedure for setting up a problem is as follows:

(1) Identify a repeating element of the structure.

(2) Associate with each flat plate element a unique number I and corresponding width B(I). (Curved segments may be modeled with several flat elements.)

2
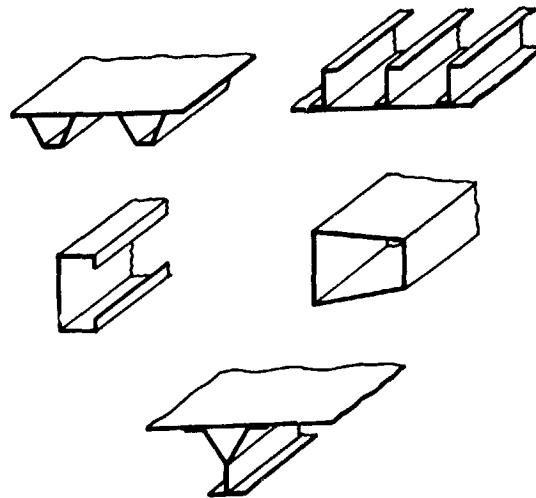
Figure 1.- Typical prismatic structures that can be
analyzed and sized with PASCO.

(3) Associate with each unique ply a number J. A ply is
described by the following four input quantities: thickness,
T(J); ply angle, THET(J); material, MAT(J,L); and temperature,
TEM(J,L).   (Temperature and material properties may be different
for each load case L.)

(4) Identify each unique combination of plies that form a
wall.   The stacking sequence for a wall is prescribed with the
input quantity KWALL(K,IW).

(5) Specify wall for each plate element by using input
quantity IWALL.

(6) Specify properties of each material in namelist MATEᴋ.

(7) Create the geometry of the repeating element through
HCARD input.

(8) Specify number of bays with the input quantity ᴎOBAY.

(9) Define the loading on the panel: NX(L), NXY(L), NY(L),
MX(L), PRESS(L).

(10) Define the overall bow on the panel: ECC(L).

(11) Define the frequency requirement: FREQ(L).

(12) Sizing variables are B(I), T(J), and THET(J). Specify active sizing variables by inputting B, T, and THET as negative numbers.

(13) Specify number of sizing cycles with the input quantity MAXJJJ. (MAXJJJ = 0 gives analysis only.)

The above constitutes the minimum input required to handle a large number of cases.

A complete set of input data and definitions is given in the section entitled COMPUTER PROGRAM DEFINITIONS AND ORGANIZATION. Much of the input is optional to give more flexibility to the user. Input that is essential is identified with an asterisk on the left side of the page in the section entitled Input. It is recommended that in the early stages of using the program, only essential input data be used to run problems. Such an approach requires only the thirteen basic steps listed above.

Several aspects of modeling, including the input quantities KWALL, IWALL, HCARD, and ICARD, and two types of linking are discussed in the following sections.

Basic Definitions

Plate.- Each flat element defined by a width B(I) and associated wall IWALL(I) is defined as plate I. There are four degrees of freedom at each edge (three displacements u, v,

4

w, and a rotation θ), so that an eight-by-eight stiffness matrix describes the structural characteristics of the plate.

Substructures.- Substructuring is used to increase the efficiency of the analysis and to simplify the modeling of complicated configurations. Substructures are created with HCARD input, which is discussed in subsequent sections and illustrated with several examples in the final section of this report. Two types of substructures can be created: doubly-connected substructures and singly-connected substructures.

• Double-connected substructure - A doubly-connected substructure is any assembly of plates and substructures with internal degrees of freedom removed and with only a beginning and final node. A plate is a special case of a doubly-connected substructure. Since a doubly-connected substructure has two nodes, its response is governed by an eight-by-eight stiffness matrix. Plates and doubly-connected substructures are numbered 1 to 120 and have an initial edge and a final edge that must be properly accounted for when assembling the final structure. The original orientation of a plate is horizontal with the initial edge on the left.

• Singly-connected substructure - If boundary conditions are known at one edge of a substructure, those degrees of freedom may be removed and only a four-by-four stiffness matrix is required. Such a substructure is denoted a singly-connected substructure. It is essential to understand that the degrees of freedom on the initial edge are the ones removed and that only the final edge is

5

available for attachment to other nodes. Singly-connected substructures are identified by numbers 121 to 899.

KWALL and IWALL Input

The input quantity KWALL(K,IW) is a sequence of integers indicating the stacking sequence for wall number IW. The sequence begins with the outside layer. Since all walls in PASCO are symmetric, only one-half the wall is specified. Each integer is a ply number which defines a thickness, orientation angle, temperature, and material. A layer with a negative orientation angle is specified with a negative integer. For example:

$$KWALL(1,3) = 5, -5, 2,$$

means that wall number 3 has a stacking sequence of

$$(THET(5), - THET(5), THET(2))_s$$

with associated thicknesses

$$(T(5), T(5), T(2))_s$$

temperatures

$$(TEM(5,L), TEM(5,L), TEM(2,L))_s$$

and materials

$$(MAT(5,L), MAT(5,L), MAT(2,L))_s$$

where L is the load case number.

The input quantity IWALL(I) is a sequence of integers indicating the wall number for plate element I. For example, assume that a stiffened panel is modeled with 7 plate elements having a total of 3 unique wall constructions. Assume that the pertinent input data are

KWALL(1,1) = 1, -1, 2, 4,

KWALL(1,2) = 3, 6,

KWALL(1,3) = 5, -5, 2,

IWALL(I) = 1, 1, 2, 1, 3, 1, 2,

The three KWALL vectors define the three unique walls. The IWALL vector means that plate elements 1, 2, 4, and 6 have wall 1; plate elements 3 and 7 have wall 2; and plate element 5 has wall 3.

Specifying only the unique walls with KWALL and IWALL decreases storage requirements and run time.

Boundary Conditions

As explained in reference 1, various boundary conditions can be specified on the lateral edges of a stiffened panel, but boundary conditions cannot be specified on the ends of a panel. The simplest way to apply boundary conditions on the lateral edges of a stiffened panel is to use the input quantity IBC. The boundary conditions available with IBC are presented in table 1. When IBC is used, the same boundary conditions are applied to both the initial edge and the final edge of the panel.

TABLE 1.- BUCKLING BOUNDARY CONDITIONS ON LATERAL EDGES
OF PANEL PRESCRIBED BY IBC

IBC	Boundary Condition Name	Restraint on rotation and displacement
0	Free	θ, w, v, u are unrestrained
1	Simple Support	θ and v are unrestrained, w = u = 0 ($\pm$909.0900)
2	Symmetry	w and u are unrestrained, θ = v = 0 ($\pm$990,9000)
3	Clamped	θ = w = v = u = 0 ($\pm$999,9900)

If IBC input is inadequate, HCARD or ICARD input can be used
to prescribe boundary conditions at any node.   Boundary conditions
are entered with two integers as $\pm$9XX,XX00 where the four X's
indicate restraint against (1) rotation θ, (2) displacement w,
(3) displacement v, and (4) displacement u, respectively.
The displacements u, v, and w are in the overall panel
coordinate system shown in figure 2.   An X = 0 means no restraint,
and an X = 9 means an infinite restraint.   See examples in
Table 1.   Without any specification the node is unrestrained,
which corresponds to a free edge boundary condition.   It is,
therefore, not necessary to specify a free edge by making all X's
equal to 0.   In fact, such a specification causes a program
error.

Examples presented in the sections entitled HCARD Input and
ICARD Input illustrate how boundary conditions can be applied.
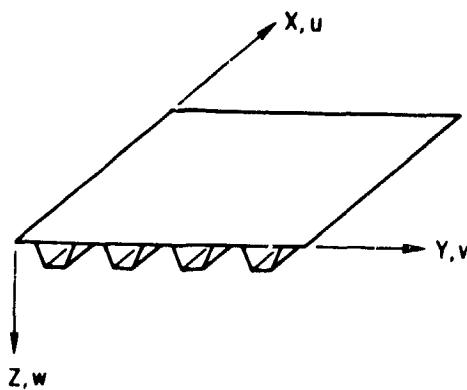
Figure 2.- Axes and displacements in overall panel
coordinate system.

HCARD Input

Whereas KWALL and IWALL input are used to specify the wall

construction for each plate element, HCARD input is used to

manipulate these plate elements to create the geometry of the cross

section. If ICARD input is used, all substructures used in

ICARD input must be defined by HCARD input.

The HCARD input is a continuous vector of integers that

are separated into groups, or sequences, that correspond in meaning

very closely to the original HCARD input to VIPASA (ref. 3). In

VIPASA, each sequence is on a separate card.

With HCARD input, new plates or substructures can be defined

from those previously generated by (1) rotating a previously

defined plate or substructure, (2) offsetting nodes associated with

initial and final edges of a previously defined plate or substruc-

ture, and (3) joining previously defined plates or substructures.

Denote the numbers on an HCARD sequence by i_1, $\pm i_2$, $\pm i_3$,

. . . $\pm i_n$. Then

i_1 Number of additional numbers in the sequence

i_2 Number of plate or substructure being generated.

i_3 Number of previously defined plate or substructure that is being modified or joined to another plate or substructure.

i_n $n > 3$. Additional instructions involving rotation of elements, offsets, joining a series of plate elements or substructures, and boundary conditions. These topics are discussed in the following sections.

It is essential that the last HCARD sequence generate the geometry of the repeating element.

Rotation.- If the orientation of a plate appearing in the final structure is not horizontal with initial edge on the left, it must be rotated to its final position. Rotation is accomplished by the sequence

$$HCARD = 4, -i_2, i_3, \pm i_4, \pm i_5, \tag{1}$$

where the initial integer 4 means there are 4 more integers making up the sequence, and i_j are all positive integers. A new plate or substructure number i_2 is generated by rotating plate or substructure i_3 through the angle which is defined by i_4 and i_5.

If i_4 is greater than 50, θ is given as in VIPASA by

$$\theta = \pm i_4 \times 10^{\pm i_5} \tag{2}$$

10

For example, HCARD = 4, -5, 1, 90, 0, means rotate element 1 by +90° and rename it element 5. This example is shown in figure 3. As another example, HCARD = 4, -6, 3, 450, -1, means rotate element 3 by +45° and rename it element 6.

HCARD = 4, -5, 1, 90, 0,
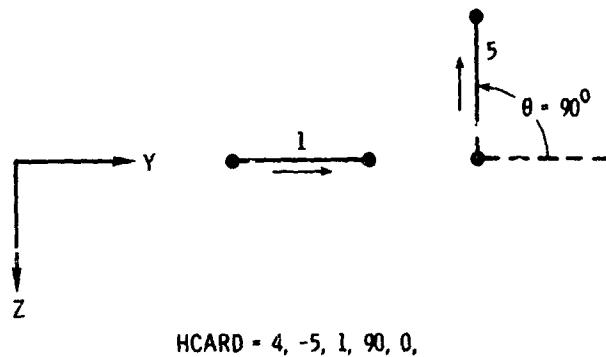
Figure 3.- Rotation with HCARD input. Arrow points from initial edge of plate element to final edge.

If i_4 is not greater than 50, θ is given by

$$\theta = \cos^{-1} \frac{\pm B(i_4)}{\pm B(i_5)} + 180°\delta \tag{3}$$

where the angle defined by $\cos^{-1} \frac{\pm B(i_4)}{\pm B(i_5)}$ is between 0° and 180°,

and

$\delta = 0$ if sign before i_4 is positive

$\delta = 1$ if sign before i_4 is negative.

For example, HCARD = 4, -13, 10, -5, 8, means rotate element 10 by the angle

11

$$\theta = \cos^{-1} \frac{-B(5)}{B(8)} + 180^{\circ}$$

and rename it element 13.

Nodes.- The junction or attachment points of plate elements and substructures are called nodes. In addition, boundary conditions are applied at nodes, and plots of buckle mode shapes are based on deflections and rotations at nodes. When first defined, each plate element has two nodes—one node at the initial edge and one node at the final edge of the plate element. These two nodes are indicated by the solid circular symbols at the edges of the plate elements in figure 3. Nodes also exist at the initial and final edges of a doubly-connected substructure and at the final edge of a singly-connected substructure.

Nodes can be moved from the edges of plate elements and substructures, and they can be eliminated. Moving a node from the edges of plate elements and substructures is referred to as offsetting the nodes. Offsetting is discussed in the next section. Nodes are eliminated by making a plate or substructure part of a larger substructure. Eliminating nodes is discussed in a subsequent section entitled Joining plates or substructures.

Offsets.- Conventional modeling, where plate element center-lines are connected, results in overlapping areas or missing areas. Offsetting nodes from the edges of plates provides a more accurate model of the intersection of two or more plates. Also, offsets allow unsymmetric laminates to be created by stacking symmetric laminates. See, for example, figure 7.

12

The nodes of a plate or substructure i_3 can be displaced from its end points to create a new plate i_2 by the following sequence.

$$HCARD = 6, \ -i_2, \ -i_3, \ \pm i_4, \ \pm i_5, \ \pm i_6, \ \pm i_7, \qquad (4)$$

The offsets in the y and z direction at the initial edge are $\pm T(i_4)$, and $\pm T(i_5)$, respectively, and for the final edge are $\pm T(i_6)$ and $\pm T(i_7)$, respectively. The sign convention is such that if the y-z coordinate system is placed at the center of the plate edge, the coordinates of the node are the proper values of the offsets.

For example,

$$HCARD = 6, \ -8, \ -1, \ -5, \ 0, \ -6, \ 7,$$

creates a new plate 8 from old plate 1 with the offsets shown in figure 4. Note that the integer zero produces a zero offset.
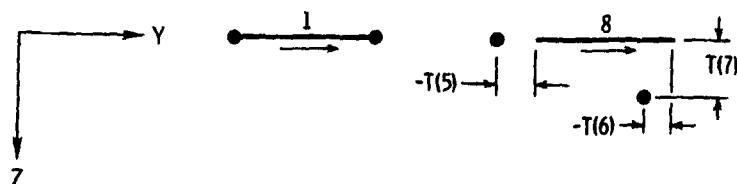
HCARD = 6, -8, -1, -5, 0, -6, 7,

Figure 4.- Offsetting nodes with HCARD input. Arrow points from initial edge of plate element to final edge.

13

The procedure used to offset the nodes at the edges of a doubly-connected substructure composed of many plate elements is the same as that used for the single plate element shown in figure 4. For a singly-connected plate or substructure, the only node is located at the final edge. Offsets for that node are defined by $\pm i_4$ and $\pm i_5$, <u>not</u> $\pm i_6$ and $\pm i_7$, although entries for i_6 and i_7 must be included in the sequence.

<u>Joining plates or substructures</u>.- Joining plates or substructures means taking previously defined plates and substructures and joining them to create the desired cross section. As explained earlier, plate elements and substructures are joined at the nodes associated with the edges of the plate elements and substructures. For example, if plate elements and substructures are defined as shown in figure 5(a), they can be joined to form the cross section shown in Figure 5(b). Nodes are shown at the edges of each plate element and substructure. Arrows point from the initial edge to the final edge of each plate element or substructure.

Whereas HCARD input specifying rotations or offsets requires a specific format (eqs. (1) and (4)), HCARD input which specifies the joining of plate elements or substructures is relatively flexible in its format. The general format is

$$HCARD = i_1, \ i_2, \ i_3, \ \pm i_4, \ \pm i_5, \ \cdot \ \cdot \ \cdot \qquad (5)$$

where i_2 and i_3 are positive. The rules for using HCARD input to join plates and substructures are stated briefly in the

14
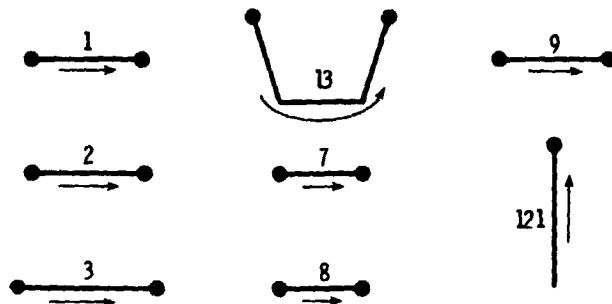
(a) Component plate elements and substructures
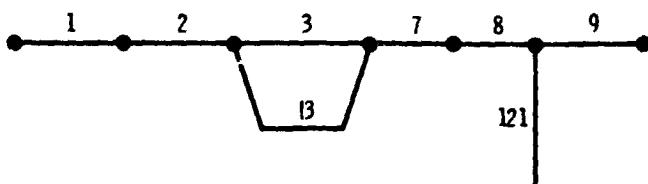
(b) Components joined to form desired cross section

Figure 5.- Plates and substructures joined to form
cross section of prismatic structure.

following paragraphs and are explained or illustrated in
subsequent examples.

If equation (5) is used to create a doubly-connected sub-
structure, the integer i_2 is 1 to 120. If equation (5) is
used to create a singly-connected substructure, the integer i_2
is 121 to 899.

Only substructures having nodes that form a simple chain can
be constructed with HCARD input. A simple chain is a simple path
that connects all the nodes.

All substructures have nodes temporarily numbered 1, 2,
3, ..., N_v. In the HCARD sequence given by equation (5), the

15

entry i_3 is the number of the first plate or doubly connected substructure that connects nodes 1 and 2. If additional plates or substructures attach at node 1 or if it is desired to apply a boundary condition at node 1, then these additional plates or substructures or these boundary conditions are all entered using negative signs—in any order. If the additional substructure is a doubly-connected substructure, it is mandatory that its geometry be such that its final node coincide with node 2 as defined by the entry i_3. If the geometry is not compatible, the program still executes, but the results are incorrect. In this case, if the plotting option is exercised, a nonfatal diagnostic is printed. Boundary conditions are entered as -9XX,XX00 where X's indicate restraint against rotation θ and displacements w, v, u, respectively. An X = 0 means no restraint and X = 9 is an infinite restraint.

After all elements connecting nodes 1 and 2 and boundary conditions and singly-connected substructures at node 1 are accounted for, the next number in the sequence is positive, indicating a plate or substructure connecting nodes 2 and 3. Again, additional negative numbers indicate other connections or boundary conditions at node 2.

This process is repeated until the substructure is described. There is a limit of 20 integers allowed in any given sequence. If more integers are required, an intermediate substructure can be defined and used to start a new sequence.

16

If a singly-connected substructure or a boundary condition is at the last node (N_V) enter a zero then the substructure or boundary condition preceded by a negative sign.

For singly-connected substructures, N_V must be greater than one, and for doubly-connected substructures, N_V must be greater than two. A diagnostic in the program is printed if this rule is violated.

Examples are now used to illustrate and explain the rules just discussed. In the examples presented in figure 6, four substructures are created from plate elements 1 to 6. Plate elements (or substructures) 5 and 6 were created earlier from plate element 1 by HCARD rotation similar to that illustrated in figure 3. The arrows on the plates and substructures point from the initial edge to the final edge. The nodes shown in the four examples in figure 6 indicate the nodes remaining after each substructure is created. Interior nodes are eliminated in HCARD modeling.

In the first example in figure 6, HCARD = 2, 121, 5, plate element 5 is changed from a doubly-connected substructure to a singly-connected substructure d¹ ioted 121. The bottom node is eliminated because the unattached ⌐ e of a singly-connected substructure must be the initial edge of that substructure.

In the second example, HCARD = 4, 7, 6, 4, 5, the initial edge of plate element 4 is joined to the final edge of plate element 6, and the initial edge of plate element 5 is joined to the final edge of plate element 4. The resulting substructure
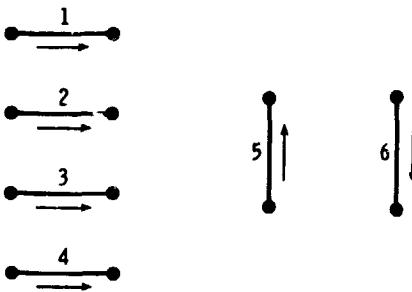
EXAMPLE 1   HCARD = 2, 121, 5,   SINGLY-CONNECTED SUBSTRUCTURE 121
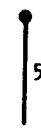
EXAMPLE 2   HCARD = 4, 7, 6, 4, 5,   DOUBLY-CONNECTED SUBSTRUCTURE 7
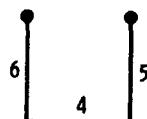
EXAMPLE 3   HCARD = 4, 8, 1, 2, -121,   DOUBLY-CONNECTED SUBSTRUCTURE 8
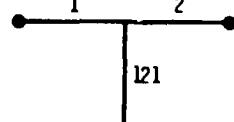
EXAMPLE 4   HCARD = 5, 9, 1, 2, -7, 3,   DOUBLY-CONNECTED SUBSTRUCTURE 9
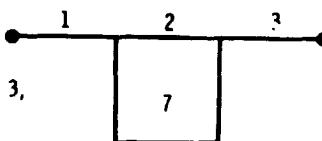
Figure 6.- Four examples of HCARD input used to join
plate elements and substructures.

is denoted 7. As shown in the figure, interior nodes are

eliminated in substructure 7. Substructure 7 is a doubly-

connected substructure.

The third example, HCARD = 4, 8, 1, 2, -121, illustrates how

two substructures can attach at the same node when one of the

substructures is singly connected. The initial edge of plate

18

element 2 is attached at the final edge of plate element 1. To attach a second substructure, here denoted 121, at the final edge of plate element 1, enter the substructure number with a minus sign. To attach additional singly- or doubly-connected substructures at the final edge of plate element 1, enter each additional substructure number with a minus sign.

The fourth example, HCARD = 5, 9, 1, 2, -7, 3, illustrates how two doubly-connected substructures can attach at the same two nodes. The initial edge of plate element 2 (a doubly-connected substructure) is attached at the final edge of plate element 1. To attach a second doubly-connected substructure, here denoted 7, at the final edge of plate element 1, enter that substructure number with a minus sign. The minus sign before the 7 indicates that a second doubly-connected substructure, here denoted 7, is also attached at the final edge of plate element 1. In this example, the rule requiring a simple chain of nodes means that doubly-connected substructure 7 must reattach at the final edge of substructure 2. This is the reason that intermediate substructure 7 was created from plate elements 4, 5, and 6.

If a singly-connected substructure is at the last node, enter a zero then the substructure number preceded by a negative sign. A more complex example which illustrates this modeling rule is presented in figure 7. The nodes for plate element 22 have been offset by one-half the sum of the thicknesses of plate elements 13 and 22. Although plate elements 13 and 22 must have symmetric laminates, the combination of plate elements 13 and 22 need not
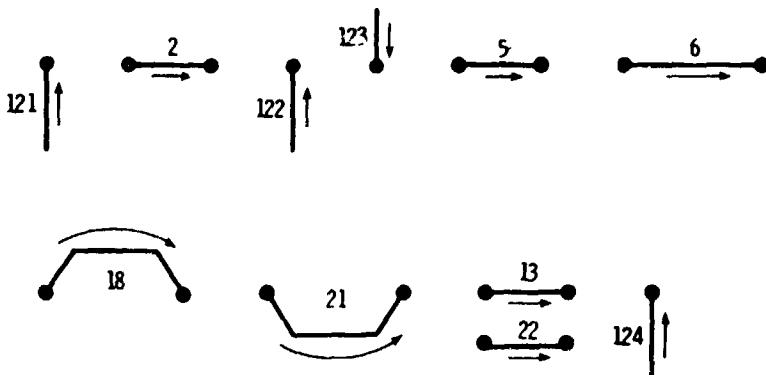
(a) PREVIOUSLY DEFINED PLATES AND SUBSTRUCTURES
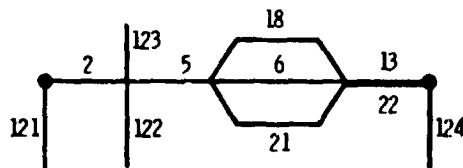
(b) SUBSTRUCTURE CREATED WITH

HCARD = 13, 23, 2, -121, 5, -122, -123, 6, -18, -21, 13, -22, 0, -124,

Figure 7.- Example of HCARD input used to create
complex substructure.

produce a symmetric laminate.   The two nodes shown in figure 7(b)
indicate the nodes remaining after doubly-connected substructure 23
is created.

Three examples illustrate how boundary conditions can be
applied with HCARD input.   In figure 8 the HCARD input

HCARD = 8, 3, 1, -909, 0900, 2, 0, -909, 0900,

places simple support boundary conditions at the node associated
with the initial edge of plate element 1 and at the node associated
with the final edge of plate element 2.   The entry 0 after

20

HCARD = 8, 3, 1, -909, 0900, 2, 0, -909, 0900,

SS △————3————△ SS
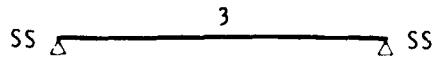
Figure 8.- Example of simple support boundary conditions
applied with HCARD input.

the 2 moves the second boundary condition to the final edae

of plate element 2.

In figure 9, the HCARD input

    HCARD = 4, 121, 5 -900, 9900,

            4, 6, 1, 2, -121,

places simple support boundary conditions at the tip of the blade.

Note the difference between the -909, 0900 in figure 8 and the

-900, 9900 in figure 9. The displacements w and v are in the

z and y directions associated with the overall structure. To

get a simple support boundary condition at the tip of the blade in

figure 9, the v and w restraints had to be reversed from

those specified in figure 8.

In figure 10, the HCARD input

    HCARD = 2, 121, 5,
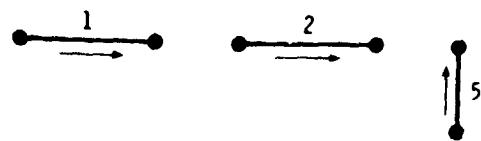
            7, 6, 1, 2, -999, 9900, 3, -121,

HCARD - 4, 121, 5, -900, 9900,

4, 6, 1, 2, -121,
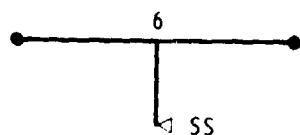
Figure 9.- Example of simple-support boundary condition
applied at tip of blade with HCARD input.
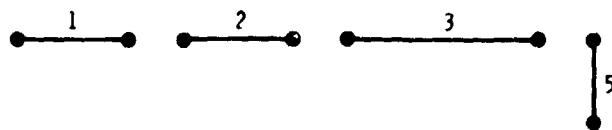
HCARD - 2, 121, 5,

7, 6, 1, 2, -999, 9900, 3, -121,
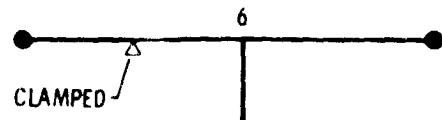
CLAMPED

Figure 10.- Example of HCARD input used to apply
boundary condition at interior node.

places clamped boundary conditions at the interior node associated

with the initial edge of plate element 2. All other edges are

free.

Finally, there is a rule which requires that there be at

least three nodes in a doubly-connected substructure and at

least two nodes in a singly-connected substructure prior to the
elimination of any node for that HCARD sequence. For example,

HCARD = 2, 3, 1,

is not legitimate because there are only two nodes, and

HCARD = 2, 122, 121,

is not legitimate because singly-connected substructure 121 has
only one node when it is used in this HCARD sequence. Also,

HCARD = 3, 9, 2, -7,

is not legitimate because substructures 2 and 7 connect the same
two nodes. A diagnostic detects this input error.

Reuse of plate and substructure numbers.- In preparing HCARD
input, the safest practice is to define new plates and sub-
structures with numbers not previously used. However, numbers
can be reused to save storage. Also, for very large problems,
it might be necessary to reuse numbers to keep the element
numbers within the maximum number of 120. Element numbers can be
reused if certain rules are followed. When a new plate or sub-
structure is defined with a number that has been previously used,
the stiffness matrix for the first plate or substructure is
destroyed so it is not available for further use. It is not
permissible to define a rotated plate or substructure with the
same number as the original plate. However, for offsets and
joining together of plates, the new plate may have the same number

23

as one of the plates from which it is constructed. Finally, additional caution is required when using numbers that are not greater than the number of plate widths B(I) input. Original plate numbers may be redefined internally to increase the efficiency of the VIPASA analysis. If any of these numbers are used for new substructures, a careful check of the HCARD data that VIPASA uses (which may be different from that input) should be made to see if the geometry is correctly defined. Plots showing the repeating element and total structure greatly facilitate this check.

ICARD Input

In most cases HCARD input is sufficient to model structures being analyzed and/or sized with PASCO. ICARD input is used only when structures cannot be modeled with HCARD input or when a more detailed mode shape plot is desired. (Mode shape plots are based on displacement and rotation at nodes, and HCARD modeling eliminates interior nodes in a repeating element.) Even when ICARD input is not used, ICARD data are generated internally and are used by VIPASA.

ICARD input is used to assemble the final structure from plates and substructures created by HCARD input. ICARD input is similar to HCARD input in that it consists of one continuous vector of integers grouped in sequences. Each sequence of integers is treated separately.

If a sequence of integers is denoted

$$\text{ICARD} = i_1, \ i_2, \ i_3, \ \ldots \ldots i_{n+2},\qquad\qquad (6)$$

then

i_1	Number of additional numbers in the sequence.
i_2	Node number. One sequence must be prepared for each node connected to a higher numbered node or having a prescribed boundary condition.
i_3	Node number (must be greater than i_2) that is connected to i_2.
i_4	Plate or substructure that connects the two nodes. Recall that each plate or substructure has an initial and final edge and it is essential that the initial edge be connected to a lower node number than the final edge.
i_5-i_n	Additional pairs of numbers, the first being the number of the node which is connected to i_2, and the second being the number of the plate or substructure making the connection. Note, if more than one element connects the two nodes, the higher numbered node number will be repeated.
i_{n+1}	Boundary conditions or the number of a singly-connected substructure at node i_2 preceded by a minus sign. Notation for boundary conditions is the same as in the HCARD ($\pm$9XX,XXOO).
i_{n+2}	Additional singly-connected substructures or a boundary condition, all entered as positive.

An example which illustrates ICARD input is presented in figure 11. The ICARD input is given by

$$\text{ICARD} = 5, \ 1, \ 2, \ 1, \ -909, \ 0900,$$
$$3, \ 2, \ 3, \ 2,$$
$$3, \ 3, \ 4, \ 3,$$
$$3, \ 4, \ 5, \ 4,$$
$$3, \ 5, \ -909, \ 0900,$$
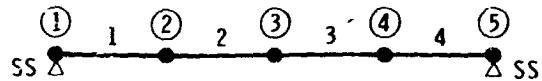
ICARD = 5, 1, 2, 1, -909,0900,
3, 2, 3, 2,
3, 3, 4, 3,
3, 4, 5, 4,
3, 5, -909, 0900,

Figure 11.- Example of ICARD input, including
input prescribing boundary conditions.

In the first sequence of integers, node 1 is connected to node 2
with plate element 1, and a boundary condition is imposed at
node 1. In the second sequence of integers, node 2 is connected
to node 3 by plate element 2. In the fifth sequence of integers,
a boundary condition is imposed at node 5. In each case, the
plate elements could have been substructures created with HCARD
input.

For the analysis, the order of the ICARD input is immaterial;
however, to obtain geometry and plotting, it is necessary that at
least one node number used in any sequence after the first must
have been used in an earlier sequence.

When ICARD input is used, it is still necessary to define a
repeating element with HCARD input in order to obtain the proper
load distribution and to obtain necessary data for panel stiffness-
es and plate centroid locations that are used in analyses involving
bending loads. It is also necessary to define the boundary

26

conditions with ICARD input (±9XX,XXOO), because the IBC parameter is not used when ICARD input is used.

The rules for ICARD input in PASCO are the same as the rules for ICARD input in VIPASA (refs. 2 to 4) with the additional capability of having to input only one repeating element of a repetitive structure. The input quantity ICREP takes the structure defined by ICARD input and repeats it ICREP times. (See section entitled NOSUB, NOBAY, and ICREP Input for specific input requirements.)

An example which illustrates ICARD input combined with HCARD input, boundary conditions, and ICREP is presented in figure 12. The plate element components and the nodes are identified at the top of the figure. (Plate elements 2, 4, 7, 9, and 11 are assumed to have been rotated with previous HCARD input to produce plate elements 14, 15, 16, 17, and 18, respectively.) The repeating element, substructure 21, is first modeled with HCARD input. ICARD input is then used to provide detailed plotting. To retain nodes 3, 4, 6, 9, and 10, basic plate elements are used rather than substructures 19, 20 and 121 created by HCARD input. Boundary conditions are imposed with ICARD input at nodes 1 and 12 of the repeating element. The input ICREP = 2 creates the final structure shown at the bottom of the figure. In the final structure, there are two repeating elements, and the boundary conditions at node 12 are shifted to the final edge of the panel. In this example, all nodes are retained in the final structure.
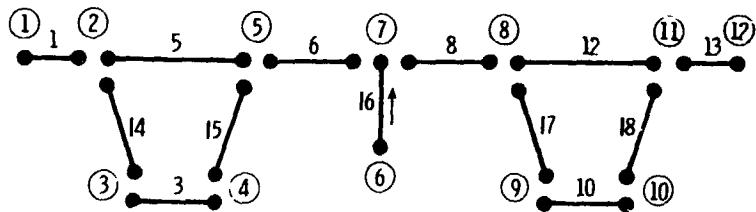
```
HCARD = 4, 19, 14, 3, 15,
        4, 20, 17, 10, 18,
        2, 121, 16,
        10, 21, 1, 5, -19, 6, 8, -121, 12, -20, 13,

ICARD = 5, 1, 2, 1, -909, 0900,
        5, 2, 5, 5, 3, 14,
        3, 3, 4, 3,
        3, 4, 5, 15,
        3, 5, 7, 6,
        3, 6, 7, 16,
        3, 7, 8, 8,
        5, 8, 11, 12, 9, 17,
        3, 9   , 10,
        3, 10, 11, 18,
        3, 11, 12, 13,
        3, 12, -909, 0900,

ICREP = 2,
```


FINAL STRUCTURE
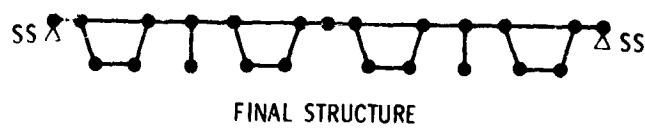
Figure 12.- Example of ICARD input, including HCARD and ICREP input and boundary conditions.

Whereas only structures and substructures having nodes that form a simple chain can be modeled with HCARD input, that restriction is not imposed on structures modeled with ICARD input.

In the example presented in figure 12, the nodal path 2, 3, 4, 5 and the nodal path 2, 5 produce a double chain that cannot be modeled with HCARD input. To model the structure with HCARD input, intermediate substructure 19 was created. In doing so, nodes 3 and 4 were eliminated. Nodes 3, 4, 6, 9, and 10 can be retained only with ICARD input. Examples which illustrate complex modeling with ICARD input are presented in a subsequent section entitled EXAMPLES.

NOSUB, NOBAY, and ICREP Input

The input quantities NOSUB, NOBAY, and ICREP are used to replicate a repeating element generated by HCARD and/or ICARD input.

HCARD requirements.- After creating a repeating element with HCARD input, an entire stiffened panel can be generated with the input NOSUB and NOBAY. The number of repeating elements in a major substructure is equal to $2^{(NOSUB-1)}$, and the number of major substructures in the final structure is equal to NOBAY. The parameter ICREP is not used when the repeating element is created only with HCARD input.

Assume that a repeating element with 1 stiffener has been defined with HCARD input. Also assume that a 16-stiffener panel is desired. The following combinations of NOSUB and NOBAY produce the 16-stiffener panel.

NOSUB	NOBAY
1	16
2	8
3	4
4	2
5	1

As a rule of thumb, a combination of NOSUB and NOBAY in which
NOSUB and NOBAY are nearly equal is recommended for initial
studies of a structure. The most efficient combination depends
upon the structure. If a 15-stiffener panel is desired, the
only combination is NOSUB = 1, NOBAY = 15.

If NOSUB and NOBAY input are used and if boundary conditions
are specified with HCARD input (-9XX,XX00), these boundary
conditions are repeated on the corresponding node of each
repeating element in the structure. An example which illustrates
NOSUB and NOBAY input is shown in figure 13.

ICARD requirements.- After creating a repeating element with
ICARD input (which also requires HCARD input), an entire stiffened
panel can be generated with the input quantities ICREP and NOBAY.
The number of repeating elements in the final structure is equal
to ICREP, which must equal NOBAY. The parameter NOSUB is not
used with ICARD input.

When using ICREP with boundary conditions, it is important
to note that boundary conditions may be applied with two types
of ICARD sequences, and that PASCO has been coded so that the type
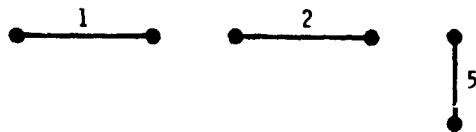
HCARD = 4, 121, 5, -900, 9900,

4, 6, 1, 2, -121,

NOSUB = 1,
NOBAY = 2,
IBC = 1,

FINAL STRUCTURE

Figure 13.- Example showing how boundary conditions applied
with HCARD input to repeating element are repeated at
corresponding nodes of final structure and how boundary
conditions applied with IBC input are applied only to the
lateral edges of the panel.

of ICARD sequence chosen will determine the location of the

boundary condition. The two types of ICARD sequences are:

(1) An ICARD sequence that contains node connections as

well as boundary conditions. (For example, 5, 1, 2, 1,

-909, 0900, in fig. 12)

(2) An ICARD sequence that contains only boundary condi-

tions - no node connections. (For example,

3, 12, -909, 0900, in fig. 12)

Boundary conditions prescribed with the first type of ICARD

sequence are applied only to the first repeating element of a

replicated structure. Boundary conditions prescribed with the

31

second type of ICARD sequence are applied only to the last repeating element of the replicated structure. No boundary conditions are applied to the interior repeating elements. In the truss-core example presented in the section entitled EXAMPLES, boundary conditions are applied with ICARD input at more than one node at each edge of the panel.

Linking

Linking equations.- It is useful to be able to specify relationships involving all the B, T, and THET variables in order to define offsets, stiffener spacing, angles, and other quantities associated with panel geometry. A set of general linear equations relating all the variables can be input with the parameters AB(I,N), AT(J,N), ATHET(J,N), and AC(N). The linear equations in terms of these parameters are given by

$$AB(I,1) \cdot B(I) + AT(J,1) \cdot T(J) + ATHET(J,1) \cdot THET(J) = AC(1)$$

$$AB(I,2) \cdot B(I) + AT(J,2) \cdot T(J) + ATHET(J,2) \cdot THET(J) = AC(2)$$

$$\ldots$$

$$\ldots$$

$$AB(I,N) \cdot B(I) + AT(J,N) \cdot T(J) + ATHET(J,N) \cdot THET(J) = AC(N)$$

$$(7)$$

Certain B, T, and THET variables in these linking equations are taken to be the dependent variables. The other B, T, and THET variables are taken to be the independent variables. Each dependent variable is input as 1.E30. A

32

consistent input then has the number of linking equations equal to the number of 1.E30 input variables. If the input is not consistent, a diagnostic message is printed.

As an example, assume that the pertinent input is

$$B(1) = -1.5, -1.5, 1.E30,$$

$$AB(1,1) = 1., 2., 1.,$$

$$AC(1) = 6.,$$

The associated linking equation is

$$B(1) + 2B(2) + B(3) = 6 \qquad\qquad (8)$$

The negative signs on B(1) and B(2) indicate that they are sizing variables. Such negative signs are changed to positive signs for the linking equations. The initial value of B(3) is calculated by equation (8) to be 1.5. Because values of B(1) and B(2) may vary during sizing, subsequent values of B(3) may not be 1.5. (See discussion in example 3.)

Automatic linking.- PASCO also contains an automatic linking feature that is applied only to plate element widths B(I) and is available when the parameter LINK is input as 1. Automatic linking is more convenient to use and saves storage and computer time relative to equivalent linking carried out with the linking equations discussed above. In general, automatic linking works in the following way: After the linking equations (eq. (7)) are applied to the variables B(I), T(J), and THET(J),

33

plate elements B(I) are organized into linking groups having
the same wall construction (same IWALL) and same width B.
These linking groups are retained throughout the sizing. During
sizing, the automatic linking feature equates the widths of all
plate elements within a group to the width of the plate element
having the lowest number in the group.

For example, assume that the pertinent input data are given
by

        LINK = 1,

        B(1) = 1.5, 3., 1.5, 1.5, 1.5,

        IWALL(1) = 1, 2, 1, 3, 1,

Plate elements 1, 3, and 5 form a linking group in which B(1)
is the controlling width and B(3) and B(5) are equated to
B(1) during sizing. Element width B(4) is not part of the
linking group because IWALL(4) = 3 rather than 1. If a plate
width is intended to be a controlling width, care should be taken
to insure that it has the lowest plate number in the group. For
example, if the pertinent input were given by

        LINK = 1,

        B(1) = 1.5, 3., -1.5, 1.5, 1.5,

        IWALL(1) = 1, 2, 1, 3, 1,

B(3) would always equal B(1) and would not act as a sizing
variable. Caution—the default value for LINK is 1.

34

If a linking group contains more than one B that is a
sizing variable, the sizing variable with the lowest plate number
is retained in the linking group, and all other sizing variables
are removed from the linking group. For example, if the
pertinent input were

    LINK = 1,

    B(1) = -1.5, 3., -1.5, 1.5, 1.5,

    IWALL(1) = 1, 2, 1, 3, 1,

the final linking group would include only B(1) and B(5),
with B(1) as the controlling width. Both B(1) and B(3)
would function as sizing variables.

Linking equations (eq. (7)) are applied to the B(I), T(J),
and THET(J) before linking groups are identified. If linking
equations cause a B(I) to be identified as part of a linking
group, and if that B(I) is not the controlling width in that
linking group, then automatic linking will override the linking
equations in subsequent calculations of that B(I). For example,
assume the pertinent input were

    LINK = 1,

    B(1) = -1.5, 3., 1.E30, -1.5, 1.5,

    IWALL = 1, 2, 1, 3, 1,

    AB(1,1) = 0., 0., -1., 1.,

The linking equation would be

$$-B(3) + B(4) = 0$$

With this input the user probably wanted B(3) to equal the
sizing variable B(4) throughout the sizing. However, because
the initial value of B(3) is calculated as 1.5, B(3) is
identified as being part of the linking group composed of B(1),
B(3) and B(5) with the sizing variable B(1) as the controlling
width. If the user wishes to work the sizing problem with
B(3) = B(4), the input for B(4) could, for example, be changed
from -1.5 to -1.505. There are, of course, many other approaches.

Load Distribution

The longitudinal (N_{x_i}), transverse (N_{y_i}), and shear (N_{xy_i})
loads in each plate element are automatically calculated based
on the applied loads, the geometry and stiffness distribution of
the cross section, and the modeling rules. The loads N_{y_i} and
N_{xy_i} in any plate element can also be prescribed using the input
quantities FNY and FNXY.

Longitudinal load, N_{x_i}.- The load distribution N_{x_i}
calculated by PASCO is the sum of a load distribution based on
uniform longitudinal strain of all elements in the panel cross
section and a load distribution caused by bending loads. The
resulting load distribution cannot be overridden with input.
Details of the stress analysis are presented in reference 1.

36

Transverse load, N_{y_i} .- In general, the modeling rules are

designed so that the full applied transverse load N_y is carried

by the skin of a stiffened panel, and no part of N_y is carried by

the stiffener elements. This objective is accomplished by the

following two rules that are applied automatically in the program.

(1) In a singly-connected substructure, such as a blade on a blade-

stiffened panel, N_{y_i} is automatically set to zero. (2) If two

or more doubly-connected substructures join the same two nodes,

N_{y_i} of the first substructure mentioned on the appropriate HCARD

sequence is automatically set equal to the full N_y, and N_{y_i} of

the second and subsequent substructures are automatically set equal

to zero. For example, if HCARD input is

        HCARD = 5, 5, 1, 2, -3, 4,

then substructures 1, 2, and 4 carry the full N_y load, and

substructure 3 carries no N_y load. The same structure could

also be modeled with the HCARD input

        HCARD = 5, 5, 1, 3, -2, 4,

In this case, substructure 3 carries the full N_y load and

substructure 2 carries no N_y load.

    If these modeling rules do not provide the desired load

distribution, the transverse loads in each plate element can be

specified with the input quantity FNY(I). The load NY(I,L)

in plate element I for load case L is given by

$$NY(I,L) = FNY(I) \cdot NY(L)$$

in which NY(L) is the applied transverse load for load case L.

If FNY(I) = 0, NY(I) is calculated according to modeling rules. Therefore, if a zero value of NY(I) is desired for plate element I and if the modeling rules give a nonzero value, enter FNY(I) as a very small number.

Shear load, N_{xy_i} .- The modeling rules in PASCO automatically specify that a singly-connected substructure carries no shear load. If two or more doubly-connected substructures join the same two nodes, the shear load is distributed between these doubly-connected substructures according to the stiffnesses of the shear paths. Details of the stress analysis are presented in reference 1.

Just as in the case of transverse loads, the load NXY(I,L) in plate element I for load case L can be specified by

$$NXY(I,L) = FNXY(I) \cdot NXY(L)$$

where NXY(L) is the applied shear load for load case L.   If FNXY(I) = 0, NXY(I) is calculated according to modeling rules.

PLOTTING

Plotting is controlled by the parameters AMP, IP, and PAGE which are defined in a subsequent section denoted Input.

When plots are obtained, special attention should be given to the first two plots to verify the correctness of the model. The first plot shows the repeating element with individual layer

38

thicknesses and element offsets. The second plot shows the geometry of each plate centerline for the complete structure. The first and second plots are generated independently. The second plot, rather than the first plot, represents the geometry actually analyzed by VIPASA.

The smeared orthotropic stiffnesses D_{11} and D_{33} are calculated within PASCO using the same information as that used to generate the plot of the repeating element. For that reason, if D_{11} and D_{33} are to be calculated correctly and if subsequent analyses based on D_{11} and D_{33} are to be correct, it is generally necessary for the repeating element shown on the first plot to be correct. Analyses that use D_{11} and D_{33} include analyses involving an N_{xy} loading with SHEAR $\neq$ 0, a lateral pressure loading, a bow-type initial imperfection, temperature with ITHERM = 1, and sizing problems involving upper or lower limits on D_{11}.

If plate numbers are reused or if the same number is used for more than one plate, some portion of the repeating element may be missing. In the case of an open section, such as a blade-stiffened panel, if the Z coordinate is correct for each plate element shown in the repeating element and if the complete structure shown in the second plot is correct, then D_{11} and D_{33} are calculated correctly and all analyses are correct. In the case of a closed section, such as a hat-stiffened panel, all plate elements making up the closed section must be shown correctly for the stiffness D_{33} to be calculated correctly.

39

Analyses that do not use D_{11} or D_{33} do not require a correct repeating element plot.

The circular symbols in the second plot indicate node locations in the final structure. Mode shapes, which are shown in subsequent plots, are interpolated from displacement and rotations at these nodes. For a plot to show complex and detailed buckling behavior, it may be necessary to introduce a large number of nodes through ICARD input. It is emphasized, however, that additional nodes do not increase the accuracy of any PASCO eigenvalue analysis.

For shear and/or anisotropy, two mode shapes are given for each eigenvalue corresponding to f_1 and f_2 in equation (37), reference 1.

The final plot shows FACTOR as a function of half-wavelength λ.

COMPUTER PROGRAM DEFINITIONS AND ORGANIZATION

In this section, the computer program is described in terms of input, output, and program organization.

Input

PASCO input is in namelist form with one title card. The order of the input is as follows:

Title Card

Namelist CONDAT

Namelist PANEL

Namelist MATER

40

Input that is considered to be essential is identified by an asterisk to the left of the FORTRAN name.

Namelist CONDAT.- Namelist CONDAT contains many of the input parameters normally used by the optimization code CONMIN (refs. 5 and 6). Parameters that normally appear in CONMIN input data but that are not included in namelist CONDAT are X, VLB, VUB, ISC, NFDG, FDCH, FDCHM, NDV, NCON, NSIDE, ICNDIR, CT, CTMIN, THETA and N1 to N5. Most of these quantities are either generated internally based on other input data or are fixed because of the approximate analysis approach. For example, ISC(N2) is set equal to 1 internally because the Taylor series expansion of the constraints used by CONMIN is always linear in the sizing variables.

All of the CONDAT variables which can be input have default values that handle normal problems, so it is only necessary to input the namelist card. Variables that can be input are defined as follows:

ABOBJ1
Fractional change in the objective function attempted during initial step in first one-dimensional search. May change during optimization. Used for interpolation. Default is 0.1.

ALPHAX
Maximum fractional change in any X(I) attempted during initial step in first one-dimensional search. May change during optimization. Used for interpolation. Default is 0.1.

41

CTL	Initial constraint tolerance to identify active constraints. IF $G(J) \geq CTL$ the constraint is defined as active. CTL is a negative number. During an optimization the absolute value of CTL is reduced. The minimum value of the absolute value of CTL is CTLMIN. Default for CTL is -0.1.

CTLMIN	Constraint tolerance to identify violated constraints. If $G(J) > CTLMIN$ the constraint is defined as violated. CTLMIN is a positive number. Default is 0.000001.

DABFUN	Convergence tolerance on absolute change in the objective function (OBJ). If in ITRM consecutive iterations $ABS(OBJ(J) - OBJ(J-1)) < DABFUN$ and no constraints are violated, the minimization process is terminated. If constraints are violated, five iterations are required to terminate. Default is 0.000001. (This is a convergence test on a single sizing cycle, not a convergence test on PASCO.)

DELFUN	Convergence tolerance. Same as DABFUN except comparison is on relative change in objective function $ABS(1.0 - OBJ(J-1)/OBJ(J))$ instead of absolute change. Default is 0.00001.

IPRINT	Print control for CONMIN. Default value is 1.
0;	Print nothing.
1;	Pring initial and final function information.
2;	Print all of the above plus control parameters. Print function values and X-vector at each iteration within CONMIN.
3-5;	Increasing amounts of print, used primarily for debugging.

ITMAX	Maximum number of optimization iterations performed by CONMIN during each sizing cycle. Default value is 15.

ITRM	Number of consecutive iterations to indicate convergence by DABFUN or DELFUN. Default is 8.

LINOBJ   Linear objective function identifier.
          If the objective function, OBJ, is known
          to be a strictly linear function of the
          sizing variables, X(I), set LINOBJ = 1.
          If OBJ is not a linear function of X(I),
          use the default value, o.

NSCAL   Scaling control parameter.

          NSCAL.EQ.0;   Do not scale the sizing
                         variables.

          NSCAL.GT.0;   Scale the sizing variable
                         every NSCAL iterations.
                         Variables are normalized
                         so that scaled X(I) = X(I)/
                         ABS(X(I)).

                         When using the scaling
                         option, it is desirable
                         that NSCAL = NDV+1 where
                         NDV is the number of
                         sizing variables. Default
                         is 0.

Namelist PANEL.- Namelist PANEL provides information

concerning the initial panel configuration, design conditions,

sizing variables, and boundary conditions. The integer sub-

scripts have the following meanings:

SUBSCRIPT	REFERS TO
I	plate I
II	sizing variable II
IW	IWALL number
J	general ply number J
JJ	general integer array
K	K^{th} layer in a given wall
L	load case L
M	eigenvalue wave number in longi-tudinal direction
MA	material number MA
N	equation number N that links the sizing variables

43

The program is dimensioned to allow large enough numbers of input quantities for most practical problems. If there is need, dimensions can be increased as indicated in a subsequent section entitled Program Operation. After the data are read and processed, it is compressed into a blank common block so that core size is only that required for the current problem. All variables are defaulted to zero unless otherwise specified.

AB(I,N), AT(J,N) ATHET(J,N), AC(N)	Matrices which can be used to define a general linear relation between the variables as follows AB(I,N) · B(I) + AT(J,N) · T(J) + ATHET(J,N) · THET(J) = AC(N). These linking equations are used to calculate the values of B's, T's, and THET's that are considered to be dependent variables. Dependent variables are identified by using input values = 1.E30.
AMP	Plotting parameter that gives the amplitude, in inches, of the buckling mode shape. Default, AMP = 0.5 times PAGE.
A11L(L), A11U(L)	Lower and upper bounds, respectively, prescribed for smeared orthotropic extensional stiffness of panel.
A33L(L), A33U(L)	Lower and upper bounds, respectively, prescribed for smeared orthotropic shear stiffness of panel.
* B(I)	Width of plate element I. If input as negative, will be used as a sizing variable with positive value.
BL(I), BU(I)	Lower and upper bounds on B(I). Default value of BU(I) is 1.E10.
BMOV(I)	Governs move limits for sizing variables associated with plate widths. Move limits for each sizing cycle are generated internally and are given by

$$VLB(II) = X(II) - DVMOV(II) \cdot (SFACTR)^{n-1} \cdot X(II)_{init}$$

$$VUB(II) = X(II) + DVMOV(II) \cdot (SFACTR)^{n-1} \cdot X(II)_{init}$$

44

where:

- VLB(II) and VUB(II) are the sizing variable lower and upper bounds used by CONMIN in a sizing cycle.

- X(II) are the values of the sizing variables at the beginning of a sizing cycle.

- DVMOV(II) is equal to the appropriate corresponding element in one of the three vectors BMOV(I), TMOV(J), or THMOV(J). For example, if X(3) = B(5), then DVMOV(3) = BMOV(5). If BMOV(I), TMOV(J), or THMOV(J) are not read in, the input scalar SMOVE is used for DVMOV(II).

- SFACTR is an input scalar

- n is the sizing cycle number

- X(II)$_{init}$ are the initial (input) values of X(II).

Each move limit involves a change that is a percentage of the initial value of the associated sizing variable. One of the objectives of the move limit strategy is to reduce the move limits as the sizing progresses. Overall lower and upper bounds on the sizing variables, defined by ⁻L(I), TL(J), THETL(J) and BU(I), TU(J), THETU(J), respectively, override the lower and upper bounds VLB and VUB for a sizing cycle.

CLAM(M)   Parameter used with buckling or frequency constraints to specify margin of safety for each half-wavelength λ = EL/M. Default = 1.

$$G = 1 - \frac{F(M)}{CLAM(M)}$$

If the input parameter SHEAR is not equal to zero, orthotropic plate buckling loads are calculated and applied as a correction to the M=1 VIPASA buckling load. The adjusted shear analysis and the definition of F are discussed in reference 1.

45

CONV1	Eigenvalue convergence criterion used during eigenvalue analysis to determine important buckling half-wavelengths. Eigenvalues are calculated to an accuracy of one part in CONV1. Default CONV1 = 500.
CONV2	Eigenvalue convergence criterion. Once the important buckling wavelengths are identified using CONV1 as a relatively rough convergence criterion, eigenvalues that are to be used as constraints are calculated to an accuracy of one part in CONV2. Derivatives of these eigenvalues are also calculated with CONV2. Default = 500,000.
DELDV, DELB DELT, DELTH	Parameters used in determining incremental value of sizing variables for numerical derivatives.

$$\Delta X = DELDV \cdot X(II)_{init} + DELX$$

where

$X(II)_{init}$ are the initial (input) values of $X(II)$.

DELX = DELB if X is an element width,

DELX = DELT if X is a ply thickness, and

DELX = DELTH if X is a ply orientation angle.

DELDV, DELB, DELT, and DELTH are input quantities.

DELX is assigned the proper value internally. Default, DELDV = 0.0005.

DELM — Controls range of eigenvalues that are examined for numerical derivatives. To speed convergence of the perturbed eigenvalue solution, the range of F examined for the perturbed solution is

$$F(JJ) \pm \Delta F(JJ)$$

where

$$\Delta F(JJ) = DELM \cdot DELDV \cdot FACTOR(JJ)$$

46

and F(JJ) i; the solution for the unperturbed values of the sizing variables. (See fig. 28, ref. 1) It should be noted that ΔF varies with the finite difference increment via DELDV. However the finite difference increment also depends upon DELB, DELT, and DELTH.

Increases in DELB, DELT, or DELTH may require increases in DELM. Default, DELM = 10.

D11L(L), D11U(L)	Lower and upper bounds, respectively, prescribed for smeared orthotropic bending stiffness of panel. Stiffness resists panel bending moment M_x.
ECC(L)	Amplitude of bow-type imperfection at panel midlength for load case L.
* EL	Panel length L.
FNXY(I), FNY(I)	Optional input that can be used to override the panel internal loads N_{xy} and N_y that are calculated automatically by the program according to modeling rules. The value of N_{xy_i} or N_{y_i} in any plate i is given by

$$N_{xy_i}(L) = FNXY(I) \cdot NXY(L)$$

$$N_{y_i}(L) = FNY(I) \cdot NY(L)$$

where NXY and NY are the applied (input) loads for load case L. If a user wishes to specify a zero value of N_{xy_i} or N_{y_i} in an element, when the modeling rules give a nonzero value, he should input a very small number (1.E - 10) rather than zero.

* FREQ(L)	Frequency requirement for load case L in cycles per unit of time.
FSTIFF	Parameter used to specify half-wavelength λ used by VIPASA to evaluate stiffnesses A_{22}, A_{33}, and D_{22}. λ = FSTIFF $\cdot$ EL. Default, FSTIFF = 10.
GRANGE	Constraint deletion parameter. Indicates range of constraint values retained for Taylor series expansion and CONMIN. For lower bound constraints, the minimum value of G retained is

$$G = 1 - GRANGE$$

For upper bound constraints, the minimum value of G retained is

$$G = \frac{1}{GRANGE} - 1$$

Default, GRANGE = 2.

* HCARD(JJ)   Integer array used to create substructures and to define geometry of repeating element. The HCARD input is used to rotate plates and substructures, offset plates and substructures, and join plates and substructures. A complete discussion of HCARD input is presented in the section entitled Modeling.

IBAY   For repetitive structures described by ICARD input, IBAY is an integer which, when added to any node number, produces the correct corresponding node number in the next repetitive structure. Used only when ICREP > 1. Default is nn-1, where nn is the maximum node number in the repeating element described by the ICARD input. (nn - 1 is the correct value for IBAY if repeating elements are connected only at one node.)

* IBC   Controls boundary conditions along edges y = constant. Not used when ICARD input is used. See page 7.

= 0   free edge

= 1   simple support

= 2   symmetry

= 3   clamped

ICARD(JJ)   Optional integer array used to assemble final structure from plates and sub-structures created by HCARD. Used when more detail is desired in mode shapes or when repeating element cannot be described by HCARD. A complete discussion of ICARD input is presented in the section entitled Modeling.

ICREP	Can be used with ICARD input when structure is repetitive. The structure defined by ICARD will be repeated ICREP times.
IJB	Controls diagnostic printing. If set equal to 1, information concerning negative roots is printed during every VIPASA iteration.
* IORTH	If set equal to 0, anisotropic bending stiffnesses are ignored. If set equal to 1, anisotropic bending stiffnesses are retained. Default = 0 when $N_{xy} = 0$; =1 when $N_{xy} \neq 0$.
IP	Plot and print indicator.

= 0   no plots

= ±1   plot of undeformed structure. This value of IP is used to obtain only a stress and stiffness analysis. No buckling analysis is carried out

= ±2   plot of undeformed structure plus plot of buckling modes superimposed on undeformed structure

= ±3   plot of buckling mode only

If negative sign is used, the output includes coordinates of each plate element and substructure defined by HCARD and the coordinates of every plate appearing in the assembled structure.

IPAST	Maximum number of iterations allowed in VIPASA to determine eigenvalue for one-half wavelength. Default = 100.
ITHERM(L)	Specifies treatment of bending moment caused by temperature and transverse load. If ITHERM=1, the panel is allowed to take on a bow, the magnitude of which is calculated to produce zero bending moment in a panel loaded only by temperature and transverse load. If ITHERM=0, the panel remains flat, and the bending moment caused by temperature and transverse load is retained. See also discussion of bending loads in reference 1 and examples in reference 7.

* IWALL(I)		Wall number for plate I. See section entitled KWALL and IWALL Input. Default, IWALL(I) = I.
JDER		Derivatives of buckling constraints can be calculated with a rapid, one-iteration approximation or a complete analysis. JDER is an integer which indicates the number of the sizing cycle at which the derivative calculation changes from approximate to complete. For example, if JDER = 5 and MAXJJJ = 8, the one-iteration approach would be used for sizing cycles 1 to 4, and complete buckling analyses would be used for sizing cycles 5 to 8. Default, JDER = MAXJJJ + 1, which means that approximate derivatives are used for all sizing cycles unless JDER is specified otherwise.
JPRINT		Print indicator. The higher JPRINT, the greater the amount of printout.

1   buckling solution printed only for initial design and final design

2   buckling solution printed for every sizing cycle

3   all buckling solutions printed, including those used to determine derivatives

* KWALL(K,IW)		A sequence of integers indicating the stacking sequence for wall number IW. See section entitled KWALL and IWALL Input.
LINK		Indicates whether automatic linking is to be used for B(I). If LINK = 0, automatic linking is not used. If

(1) LINK = 1 and

(2) plate elements I and J have the same wall construction and

(3) B(J) = B(I)

then automatic linking causes B(J) to equal B(I) throughout the sizing. A complete discussion of linking is presented in the section entitled Linking. The default value of LINK is 1.

MAT(J,L)   Specifies the material number used in the Jth ply for load case L. This parameter is a function of load case to allow material properties to be different for different load cases. Default, MAT(J,1) = 1, and MAT(J,L) = MAT(J,1) for L > 1.

* MAXJJJ   Number of sizing cycles used in an overall synthesis. No convergence criteria are provided to reduce the number of sizing cycles. When MAXJJJ = 0, the code carries out only an analysis of the initial design.

MAXL   Maximum number of values of half-wavelength λ for which buckling or frequency constraints are calculated. See reference 1, section entitled Identifying critical buckling and frequency constraints. Default, MAXL = 15.

* MINLAM   Specifies smallest half-wavelength for which buckling loads are examined, EL/MINLAM. Default, MINLAM = 1.

MSEL   Selects VIPASA mode shape to be plotted. For certain problems, numerical ill-conditioning can occur in calculation of mode shapes. An attempt is made to remove this ill-conditioning by setting certain nodal displacements equal to zero. MSEL = 1 specifies mode shape with ill-conditioning removed. MSEL = 2 specifies mode shape as originally calculated. Default = 1.

* MX(L)   Applied bending moment per unit width for load case L. Should be used when loading is primarily bending and it is desired to have bending forces included in the calculation of the λ = EL buckling mode. Bending forces caused by M_x are not influenced by inplane loads. See equation (20), reference 1.

51

NEIG(M)	Number of eigenvalues determined at half-wavelength $\lambda = EL/M$. Default, NEIG = 1.
NLAM(JJ)	Specified values of M ($\lambda = EL/M$) for which buckling or frequency eigenvalues are calculated. The program automatically calculates the eigenvalue for M = 1. The program also calculates eigenvalues for other values of M based on logic discussed in reference 1, section entitled Identifying critical buckling and frequency constraints. With NLAM input, the user can specify additional values of M for which eigenvalues are calculated.
* NOBAY	Number of major substructures in final structure. Default, NOBAY = 1.
NOSUB	Number of repeating elements in a major substructure is equal to $2^{(NOSUB-1)}$. Default, NOSUB = 1.
* NX(L), NY(L), NXY(L)	Applied inplane longitudinal, transverse, and shear load per unit width of panel for load case L.
PAGE	Controls size of plot. Default = 1.
* PRESS(L)	Uniform lateral pressure loading for load case L.
SFACTR	Governs the rate at which move limits are decreased. See definition of BMOV(I). Default value of SFACTR is 0.8.
SHEAR	SHEAR $\neq$ 0 causes orthotropic plate buckling loads to be calculated and applied as a correction to the M = 1 VIPASA buckling load. The value of the twisting stiffness used in the calculation of the orthotropic plate buckling loads is the product of SHEAR and the calculated value of the twisting stiffness. See reference 1, section entitled Adjusted Analysis for Shear Buckling. See also reference 8.
SMOVE	If BMOV(I), THMOV(J), or TMOV(J) are not read in, the scalar SMOVE is used in their place. See discussion of BMOV(I). Default value of SMOVE is 0.2.

*	T(J)	Thickness of Jth ply. If input as negative, will be used as a sizing variable with a positive value.
*	TEM(J,L)	Change in temperature of ply J for load case L.
*	THET(J)	Fiber orientation angle of Jth ply. If input as negative will be used as a sizing variable with a positive value.
	THETL(J), THETU(J)	Lower and upper bounds, respectively, on THET(J). Default value of THETU(J) is 1.E10.
	THMOV(J), TMOV(J)	Governs move limits for sizing variables associated with fiber orientation angle and ply thickness. See definition of BMOV(I).
	TL(J), TU(J)	Lower and upper bounds, respectively, on T(J). Default value of TU(J) is 1.E10.

Namelist MATER.- Namelist MATER provides material properties, indicators for material strength criteria, and allowables used in material strength criteria. Subscript MA refers to material number. Any orthotropic material may be used. Special emphasis has been placed on composite materials with direction 1 as the fiber direction and direction 2 as transverse to the fiber direction.

*	E1(MA),E2(MA)	Young's modulus of orthotropic material in 1 and 2 directions, respectively.
*	E12(MA)	Inplane shear modulus of orthotropic material in 1-2 coordinate system.
*	ANU1(MA)	Poisson's ratio of orthotropic material defined by ANU1 · E2 = ANU2 · E1.
*	RHO(MA)	Mass density of material.
*	ALFA1(MA)	Coefficient of thermal expansion of orthotropic material in 1 direction.

*	ALFA2(MA)		Coefficient of thermal expansion of orthotropic material in 2 direction.
*	ALLOW(JJ,MA)		Constants used to define material failure criterion and allowables used in criterion. ALLOW(1,MA) is used to select failure criterion. A value of 1 indicates an allowable stress criterion; 2 indicates allowable mechanical strain; 3 indicates the Tsai-Wu criterion. ALLOW(2,MA)- ALLOW(7,MA) are input values of the allowables defined in the following table.

TABLE 2.- INPUT VALUES FOR ALLOW(JJ,MA)

JJ	ALLOW(JJ,MA)		
1	1 Stress	2 Strain	3 Tsai-Wu
2	σ_1 comp.	$(\varepsilon_1 - \alpha_1 \Delta T)$ comp.	σ_1 comp.
3	σ_1 ten.	$(\varepsilon_1 - \alpha_1 \Delta T)$ ten.	σ_1 ten.
4	σ_2 comp.	$(\varepsilon_2 - \alpha_2 \Delta T)$ comp.	σ_2 comp.
5	σ_2 ten.	$(\varepsilon_2 - \alpha_2 \Delta T)$ ten.	σ_2 ten.
6	τ_{12}	γ_{12}	τ_{12}
7			F_{12}

In the table, σ_1, σ_2, and τ_{12} are allowable stresses; ε_1, ε_2, and γ_{12} are allowable strains; α_1 and α_2 are coefficients of thermal expansions; ΔT is the change in temperature; F_{12} is a quantity that appears in the Tsai-Wu material strength criterion (see eq. (49), ref. 1); and ten. and comp. mean tension and compression. The allowable stresses and strains are entered as positive in compression and negative in tension.

54

Output

PASCO output begins with an organized listing of input data
and default values that define the problem to be worked. There
follows a description of each laminate wall defined by the input
KWALL. The description includes the materials, thicknesses,
orientation angles, mass, and laminate elastic properties. Then
the overall panel properties, including panel geometry, load
distribution, mass, and smeared orthotropic stiffnesses, are
presented. Results of the buckling and stress analyses are
printed in various levels of detail depending upon the print
indicators. The analysis of the initial design is then summarized
in a table which gives the values of the critical constraints and,
if structural sizing is desired, the derivatives of these con-
straints with respect to each active sizing variable. This table
contains all data used to generate the Taylor series expansions
of the constraints.

The constraint table of the initial design is followed by
CONMIN output describing the first sizing cycle. The panel
configuration obtained by CONMIN is then analyzed, and a new
constraint table is printed. The print pattern of complete
analysis and CONMIN sizing is repeated MAXJJJ times, then the
final design is defined in detail.

Several examples of PASCO output are presented in the
section entitled EXAMPLES.

Program Operation

The PASCO computer program was developed on a CDC computer. Interim versions have been made operational on IBM computers. From this experience, an attempt has been made to minimize changes required to make the program operational on IBM and other computers. In many cases, double precision is required on IBM computers or other computers with similar-length words.

Organization.- The general program organization is outlined in figure 14. There are a number of comment cards distributed throughout the source code that provide additional details regarding program organization and logic.

Dimensions.- The program is dimensioned to allow large enough numbers of input quantities for most practical problems. If there is need, dimensions can be increased (or decreased) by changing common block BL5 and associated variables that appear in PROGRAM COMMON (CDC overlay version of PASCO). Instructions for redimensioning are given in detail in the comment cards in PROGRAM COMMON. Variables that can be redimensioned are:

- Maximum number of plate elements, NOBZ
- Maximum number of layers in a given wall, IKMAXZ
- Maximum number of unique layers throughout the structure, MAXNLZ
- Maximum number of values of λ for which eigenvalues are calculated, MNLAMZ
- Maximum number of load cases, LOADZ
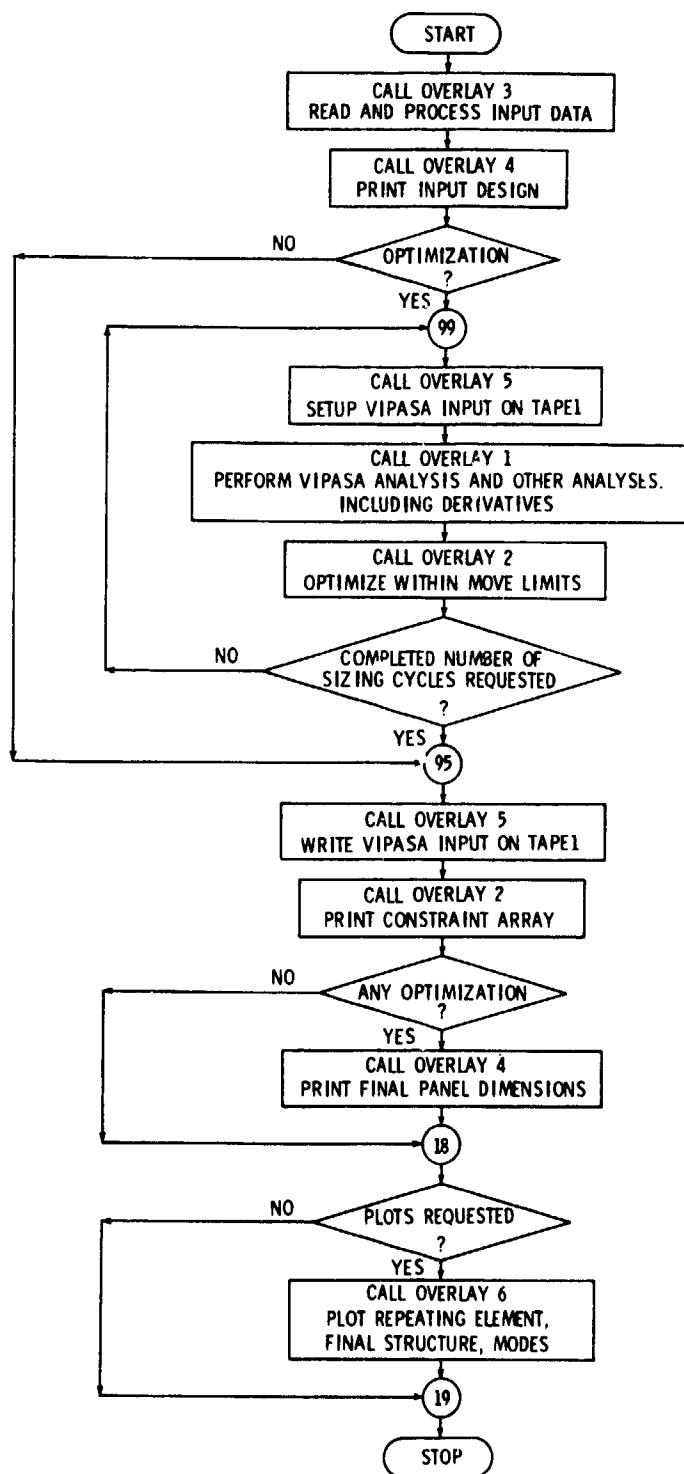
Figure 14.- PASCO program flow diagram.

- Maximum number of integers appearing in HCARD and ICARD input, JJHZ and JJIZ, respectively

- Maximum number of materials, IDUMJZ

- Maximum number of linking equations, M1Z

- Maximum number of constants used to define material strength criteria, NALLOW

Diagnostics.- An attempt has been made to provide diagnostics for common input errors that lead to inconsistent or incomplete problem definition. Undoubtedly some errors will not be detected by the diagnostics. Also, unreasonable proportions can lead to numerical ill-conditioning that causes program dumps.

Plotting.- Because individual computer centers generally have their own plotting commands, the plotting command subroutines that are used in the version of PASCO operational at Langley Research Center are not included in the version of PASCO that is made available to other users. The subroutines that execute the plotting in the Langley version of PASCO are described with comment cards in the source listing in sufficient detail that user subroutines can be substituted to perform the same functions. Additional information on plotting subroutines is given in reference 4.

EXAMPLES

Several examples are presented to illustrate in detail the features of and modeling techniques used in PASCO. For each example, the following information is presented: (1) a computer drawing of the repeating element of the initial design, (2) a computer drawing of the cross section of the entire panel with nodes, (3) a figure showing the design loading, (4) a listing of the input, and (5) selected program output. Taken as a whole, the selected output for each example encompasses the major portion of the output of the program. The numbers on each repeating element are the original plate element numbers. Units are not used in input or output quantities in the computer program and are not shown on the figures.

Example 1, Unstiffened Plate

The first example is a laminated, unstiffened plate loaded in longitudinal compression as shown in figure 15. The stacking sequence shown in figure 15(c) is created with the input T, THET, and KWALL presented in figure 16. The stacking sequence is $(+45^{\circ}, -45^{\circ}, 90^{\circ}, 0^{\circ})_s$ with corresponding thicknesses $(.02, .02, .02, .01)_s$. The negative signs on the T and THET input data indicate that the sizing variables are $T(1)$, $T(2)$, $T(3)$, $THET(2)$, and $THET(3)$. For this simple example which has only one plate element, the default value $IWALL(1)=1$ is correct. HCARD input is not required because the entire structure is one plate element.

(a) Repeating element

(b) Entire panel, with nodes
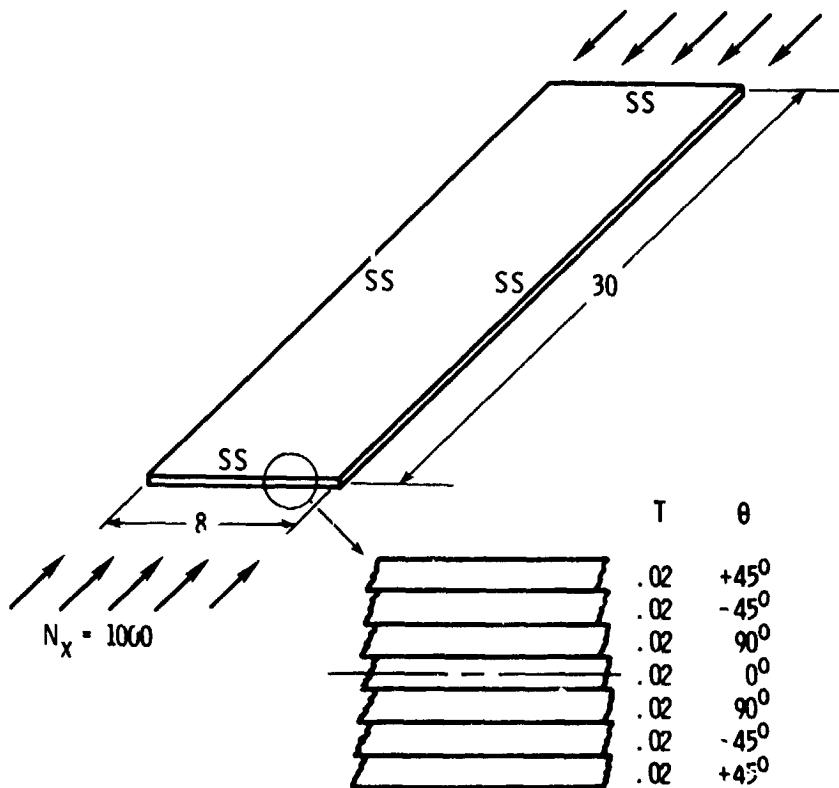
(c) Panel with design loading

Figure 15.- Example 1, unstiffened laminated plate designed
for longitudinal compression.

```
UNSTIFFENED PLATE
$CONDAT
IPRINT=2,
$
$PANEL
MAXJJJ=1,
B(1)=8.,
T(1)=-.01, -.02, -.02,
THET(1)=0., -45., -90.,
KWALL(1,1)=2,-2, 3, 1.
NX(1)=1000.,
IBC=1.
EL=30.,
IP=2,
MINLAM=30,
$
$MATER
E1(1)=19.E6, E2(1)=1.89E6, E12(1)=.93E6,   AMU1(1)=.31, RHO(1)=.0571,
ALFA1(1)=-.005E-6,     ALFA2(1)=21.8E-6,
ALLOW(1,1)=2, .004, -.004, .004, -.004, .01.
$
```

Figure 16.- Input for example 1, unstiffened laminated plate
subjected to longitudinal compression.

The output selected for this example is simply output related to the model. It describes the wall construction and gives the wall properties for the one wall used in this example. The output is presented in figure 17. In other analysis and sizing problems, a similar set of wall property information is presented for each unique wall. In PASCO, where all walls are balanced and symmetric, ZREF is half the total thickness. The quantities El, E2, and E12, denoted LAMINATE PROPERTIES, are given by

$$E1 = \frac{A_{11} - (A_{12})^2/A_{22}}{\text{Total Thickness}}$$

$$E2 = \frac{A_{22} - (A_{12})^2/A_{11}}{\text{Total Thickness}}$$

$$E12 = \frac{A_{33}}{\text{Total Thickness}}$$

```
                    WALL    NUMBER    1

        MAT         T          THET        TEM

         1       .0200000     45.00000    0.00000
         1       .0200000    -45.00000    0.00000
         1       .0200000     90.00000    0.00000
         1       .0100000      0.00000    0.00000
         1       .0100000      0.00000    0.00000
         1       .0200000     90.00000    0.00000
         1       .0200000    -45.00000    0.00000
         1       .0200000     45.00000    0.00000

ZREF FOR WALL NO.    1=    .7000E-01

MASS PER UNIT AREA OF WALL NO.    1=    .7994E-02

    A-MATRIX FOR WALL NO.   1              D-MATRIX FOR WALL NO.   1              CIT

  .9799E+06   .4066E+06   .8382E-08    .1415E+04   .9879E+03   .3455E+03   0.
  .4066E+06   .1325E+07   .2650E-06    .9879E+03   .1703E+04   .3455E+03   0.
  .8382E-0A   .2650E-06   .4540E+06    .3455E+03   .3455E+03   .1065E+04   0.

                                  LAMINATE PROPERTIES
     E1          E2          E12         NU1          NU2         ALFA1        ALFA2
  .610831E+07  .826206E+07  .324264E+07   .3067674     .4149314   .352678E-05  .157376E-05
```

Figure 17.- Sample output for example 1, unstiffened
   laminated plate designed for longitudinal compression.
   Output gives wall property information.

Example 2, Blade-Stiffened Panel

The second example is a blade-stiffened panel loaded in longitudinal compression as shown in figure 18.

The input data T, THET, and KWALL shown in figure 19 are used to create three walls. For example, wall 3 has a stacking sequence of $(+45^O, -45^O, -45^O, +45^O, 0^O)_s$ with corresponding thicknesses $(.004, .004, .004, .004, .04)_s$. The input IWALL(I) assigns the wall number of plate element I. For example, plate element 4 has wall 2.

The panel geometry is modeled with one line of HCARD data consisting of three data sequences. The input NOSUB=3 produces four repeating elements per major substructure or bay and determines the spacing of the nodes. The input NOBAY=4 produces four of these bays for the complete panel. The panel, therefore, has sixteen stiffeners.

The negative signs on B(1) and B(2) cause the stiffener spacing to be variable during panel sizing. Since the number of stiffeners is fixed, the panel width becomes variable for this example. (Linking equations could have been used to fix B(1) + B(2).) Automatic linking is used to cause B(4) to equal B(2) and B(5) to equal B(1) during sizing.

The output chosen for this example shows a portion of the organized listing of input data and default values that define the problem. This output, which is presented in figure 20, shows the plate element widths B(I), the layer thicknesses T(J),

and the layer orientation angles THET(J), together with their
lower and upper bounds for sizing.
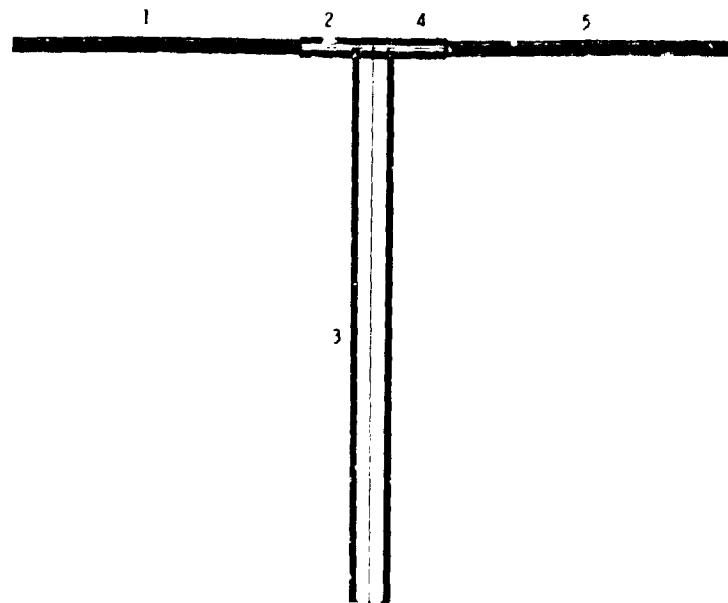
(a) Repeating element

(b) Entire panel, witn nodes
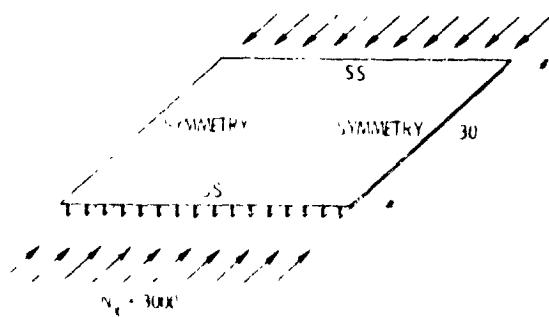
(c) Panel with design loading

Figure 18.- Example 2, blade-stiffened panel with sixteen
stiffeners, designed for longitudinal compression.

66

```
*********       BLADE-STIFFENED PANEL        ***************
$CONDAT
$
$PANEL
MAXJJJ=2,
LINK=1,
IP=2,
R(1)=-.8, -.2, -1.5, .2, .8,
T(1)= -.004, -.004, -.003, -.01, -.04,
THET(1)= 45, 45, 0, 0, 0,
KWALL(1,1)= 1,-1,-1,1,3,
KWALL(1,2) =1,-1,-1,1,4,
KWALL(1,3) = 2,-2,-2,2,5,
IWALL(1)=1,2,3,2,1,
JDFR=1.
HCARD(1)= 4,-6,3,90,0,       2,121,6.        6,7,1,2,4,-121,5,
NOBAY=4,
NOSUB=3,
IRC=2,
FL=30.,
NX(1)=3000.,
NEIG(1)=4,
NLAM(1)=1,3,17,
MINLAM=30,
$
$MATER
E1(1)=19.E6,  E2(1)=1.89E6,  E12(1)=.93E6,   ANU1(1)=.31,  RHO(1)=.0571,
ALFA1(1)=-.005E-6,      ALFA2(1)=21.8E-6,
        ALLOW(1,1)= 2, .004, -.004, .004, -.004, .01,
$
```

Figure 19.- Input for example 2, blade stiffened panel
subjected to longitudinal compression.

IN THE FOLLOWING--
ASTERISKS PRINTED FOR   B,   TL,   AND   THETL   INDICATE THE ASSOCIATED DIMENSION IS NOT A DESIGN VARIABLE

THE INTEGER SUBSCRIPTS HAVE THE FOLLOWING MEANING:
I   REFERS TO PLATE
IW   REFERS TO WALL NUMBER
J   REFERS TO GENERAL LAYER J
K   REFERS TO THE KTH LAYER IN A GIVEN WALL
L   REFERS TO A LOAD CASE L
M   REFERS TO AXIAL HALF-WAVE NUMBER
MA   REFERS TO MATERIAL MA
N   REFERS TO EQUATION NUMBER N LINKING VARIABLES

PLATE ELEMENT WIDTHS

VARIABLE NO.	B(I)	LOWER BOUND BL(I)	UPPER BOUND BU(I)	WALL NO.	VIPASA PLATE NO.	FNY(I)	FNXY(I)
1	.800000	0.000	.100000E+11	1	1	0.000000	0.000000
2	.200000	0.000	.100000E+11	2	2	0.000000	0.000000
3	1.500000	0.000	.100000E+11	3	3	0.000000	0.000000
4	.200000	*************	.100000E+11	2	2	0.000000	0.000000
5	.800000	*************	.100000E+11	1	1	0.000000	0.000000

LAYER CHARACTERISTICS

LAYER NO.	VAR NO.	THICKNESS T(J)	LOWER BOUND TL(J)	UPPER BOUND TU(J)	VAR NO.	ANGLE THET(J)	LOWER BOUND THETL(J)	UPPER BOUND THETU(J)	MATERIAL MAT(J,L)	TEMPERATURE TEM(J,L)
1	6	.004000	0.000	.100000E+11	11	45.000000	*************	.100000E+11	1	0.000
2	7	.004000	0.000	.100000E+11	12	45.000000	*************	.100000E+11	1	0.000
3	8	.003000	0.000	.100000E+11	13	0.000000	*************	.100000E+11	1	0.000
4	9	.010000	0.000	.100000E+11	14	0.000000	*************	.100000E+11	1	0.000
5	10	.040000	0.000	.100000E+11	15	0.000000	*************	.100000E+11	1	0.000

Figure 20.- Sample output for example 2, blade-stiffened panel, designed for longitudinal compression. Output identifies the variables B(I), T(J), and THET(J).

Example 3, J-Stiffened Panel

The third example is the J-stiffened panel shown in figure 21. The design loading is longitudinal compression. The panel is to be sized to carry that load assuming that the panel has either a positive bow or a negative bow. This is an example of a two-load-condition problem in which the first load condition is $N_x = 12000$, $e = +0.09$ and the second load condition is $N_x = 12000$, $e = -0.09$. These two loading conditions are illustrated in figure 21(c). The input variable for e is ECC.

Based on the input presented in figure 22, three walls are created. Each of these walls has two materials. For example, wall 3 has a stacking sequence of $(45^{\circ}, -45^{\circ}, 0^{\circ}, 90^{\circ})_s$ with corresponding thicknesses $(0.008, 0.008, 0.016, 0.008)_s$ and materials $(1, 1, 2, 1)_s$. The input quantity IWALL assigns these three walls to the fourteen plate elements.

The repeating element shown in figure 21(a) illustrates several modeling features. Offsets created with six HCARD sequences provide a more realistic model. The offsets are also shown in figure 21(b). The stiffener attachment flange is attached to the skin with short plate elements to insure that the stiffener attachment flange and the skin directly above the flange respond as a unit. For this configuration, both the attachment flange and the skin directly above the flange could have been divided into two plate elements rather than four
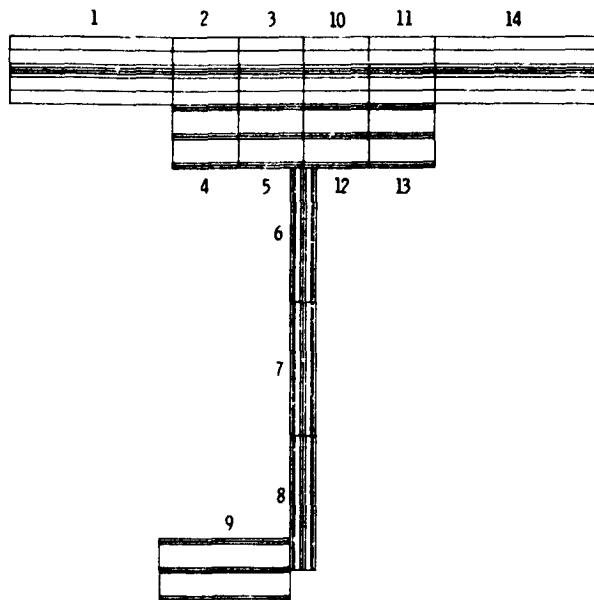
(a) Repeating element
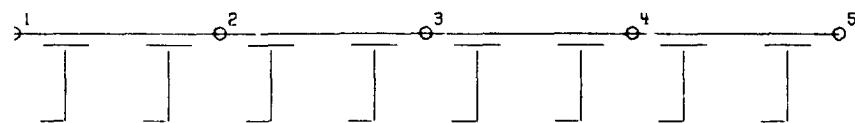
(b) Entire panel, with nodes
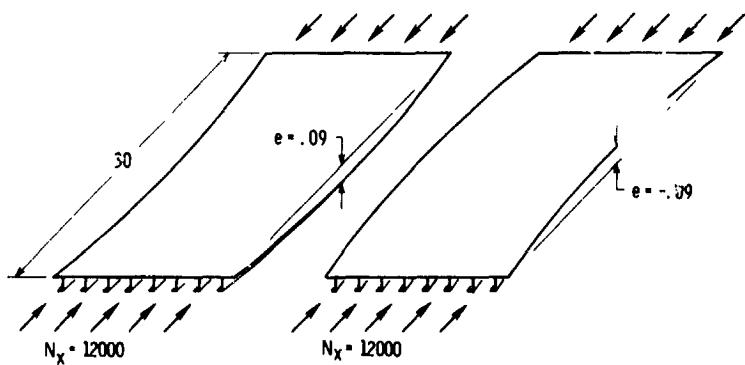
(c) Panel with design loadings

Figure 21.- Example 3, J-stiffened panel with eight
stiffeners designed for bow and longitudinal compression.

70

```
        J-STIFFENED PANEL
      $CONDAT
      $
      $PANEL
      SMOVE=.15,
      MAXJJJ=2,
      LINK=1,
      CLAM=1.3,
      JDFR=1,
      NLAM(1)=  4,10,
      NEIG(1)=4,
      MINLAM=30,
      IRC=1,
      EL=30.,
      NX(1)=12000.. 12000..
      ECC=.09, -.09,
      IP=2,
      THFT(1)=45.,0.,90.,45.,0.,90.,0.,
      MAT(1,1)=1,2,1,1,2,1,2,
      KWALL(1,1)=1,-1,2,3,
      KWALL(1,2)=4,-4,5,6,
      KWALL(1,3)=4,-4,7,6,
      IWALL=1,1,1,2,2,3,3,3,2,1,1,2,2,1,
      HCARD=   6,-15,-4,0,-8,0,-8,
               6,-16,-5,0,-8,0,-8,
               5,17,2,-15,3,-16,
               6,-18,-12,0,-8,0,-8,
               6,-19,-13,0,-8,0,-8,
               5,20,10,-18,11,-19,
               4,21,8,7,6,
               4,-22,21,90,0,
               6,-23,-22,0,0,0,-9,
               7,121,9,
               6,-122,-121,10,0,0,0,
               3,123,23,-122,
               6,15,1,17,20,-123,14,
      NORAY=4,   NOSUB=2,
      AT(1,1)=2.,1.,1.,2.,1.,1.,0.,-1.,
      AT(1,2)=2.,1.,1.,4.,2.,2.,0.,0.,-1.,
      AB(1,3)=0.,1.,0.,  -1.,
      AT(1,4)=0.,0.,0.,2.,0.,1.,1.,0.,0.,-1.,
      A11L(1)=2200000.,   A33L(1)=450000.,
      T(1)=-.04, -.012, -.008, -.008, -.07, -.008, -.016,    3*1.E30,
      B(1)=-.5, -.2, .2, 1.E30, .2, -.4, .4, .4, -.4, .2, .2, .2, .2, .5,
      $
      $MATER
      E1(1)=11.E6, E2(1)=.8E6, E12(1)=.3E6, ANU1(1)=.34, RHO(1)=.05,
      ALFA1(1)=.1E-98, ALFA2(1)=1.E-5,
      ALLOW(1,1)=2,.003,-.017,.03;-.005,.01,
      E1(2)=32.E6, E2(2)=2.7E6, E12(2)=.93E6, ANU1(2)=.21, RHO(2)=.072,
      ALFA1(2)=.24E-5,  ALFA2(2)=1.15E-5,
      ALLOW(1,2)=2,.009,-.005,.007,-.003,.010,
      $
```

Figure 22.- Input for example 3, J-stiffened panel with eight
stiffeners, designed for bow and longitudinal compression.

plate elements.[1] Because of the bow, the web of the stiffener

is modeled with three plate elements to allow the stress in the web

to vary as the panel bends. (See fig. 8, ref. 1.)

When a panel is to be sized to account for a bow, the input

CLAM(1) should be used to specify a margin on the M=1 buckling

load. The strategy is to use a combination of CLAM(1), SMOVE,

and SFACTR that prevents large moves when the constraints are

highly nonlinear, as they can be when F(1) approaches 1.0.

(See discussion of bending loads in ref. 1.) It is also advisable

to use the input NEIG(1) to obtain more than one eigenvalue for

the M=1 mode.

This example also illustrates the input for linking equations

(AT and AB) and stiffness requirements (A11L and A33L).

The output selected for this example gives the table of

linking equations, figure 23(a), and the plate forces and stresses

for the first load condition , ECC=0.09, figure 23(b). Each

column in the table of linking equations represents a linking

equation. The entries above the top row of asterisks are the

coefficients of the unknowns, the dependent variables. The

entries between the two rows of asterisks are the coefficients

---

[1]CAUTION: Plate elements having narrow widths appear to
cause numerical instabilities in VIPASA. As a rule of thumb,
width-to-thickness ratios should be greater than two. If
unexpected dumps occur, try remodeling with greater widths.
Plate elements having very long lengths EL can also cause
numerical instabilities. This problem can usually be solved
by setting FSTIFF equal to 1.0.

```
VARIABLF
NUMBER              LINKING EQUATION NO.
            1           2           3           4
    4    0.0000      0.0000     -1.0000      0.0000
   22   -1.0000      0.0000      0.0000      0.0000
   23    0.0000     -1.0000      0.0000      0.0000
   24    0.0000      0.0000      0.0000     -1.0000
**************************************************************
    2    0.0000      0.0000      1.0000      0.0000
   15    2.0000      2.0000      0.0000      0.0000
   16    1.0000      1.0000      0.0000      0.0000
   17    1.0000      1.0000      0.0000      0.0000
   18    2.0000      4.0000      0.0000      2.0000
   19    1.0000      2.0000      0.0000      0.0000
   20    1.0000      2.0000      0.0000      1.0000
   21    0.0000      0.0000      0.0000      1.0000
**************************************************************
   AC    0.0000      0.0000      0.0000      0.0000
```

(a) Table of linking equations

Figure 23.- Sample output for example 3, J-stiffened panel
   with eight stiffeners, designed for bow and longitudinal
   compression.

73

PLATE FORCES INCLUDING LATERAL PRESSURE AND IMPERFECTION, BENDING MOMENT = 3333.87606025

EVALUATED AT FACTOR= 1.000000

PLATE NO.	WALL NO.	BREADTH	NTV	NLV	NSV	B*NLV
1	-1	.50000	0.00	3743.97	0.00	1871.98
2	-1	.20000	0.00	3743.97	0.00	748.79
3	-2	.20000	0.00	14464.87	0.00	2892.97
4	-3	.40000	0.00	2859.60	0.00	1143.84
5	-3	.40000	0.00	1947.93	0.00	779.17
6	-3	.40000	0.00	1036.26	0.00	414.50
7	-?	.40000	0.00	2378.62	0.00	951.45

PLATE STRAINS AND STRESSES

PLATE NO	LAYER NO	EP1	EP2	EP12	SIG1	SIG2	SIG12
1	1	.723312E-03	.723312E-03	-.567152E-02	.822230E+04	.781965E+03	-.170146E+04
1	2	-.359907E-02	-.211245E-02	0.	.113113E+06	-.369938E+04	0.
1	3	-.211245E-02	.355907E-02	-.194547E-16	-.224577E+05	.229194E+04	.583642E-11
2	1	.723312E-03	.723312E-03	-.567152E-02	.822230E+04	.781965E+03	-.170146E+04
2	2	.355907E-02	-.211245E-02	0.	.113113E+06	-.369938E+04	0.
2	3	-.211245E-02	.355907E-02	.194547E-16	-.224577E+05	.229194E+04	.583642E-11
3	4	-.116819E-02	.116819E-02	-.398934E-02	-.132795E+05	.126292E+04	-.119680E+04
3	5	.316286E-02	-.316286E-02	0.	.101119E+06	-.439782E+03	0.
3	6	-.826477E-03	.826477E-03	-.136844E-16	-.830074E+04	.232504E+04	-.410533E-11
4	4	.894896E-03	.894896E-03	-.333505E-02	.101728E+05	.967463E+03	-.100052E+04
4	7	-.256242E-02	-.772629E-03	-.114401E-16	.818640E+05	-.635570E+03	0.
4	5	-.772629E-03	.256242E-02	-.227180E-02	-.786809E+04	.185538E+04	.343202E-11
5	5	-.609594E-03	.609594E-03	0.	.692960E+04	.659025E+03	-.681540E+03
5	7	-.174549E-02	.526307E-03	0.	.557649E+05	-.432944E+03	0.
6	4	-.526307E-03	.174549E-02	-.779284E-17	-.535966E+04	.126387E+04	-.233785E-11
6	7	.324292E-03	-.324292E-03	-.120855E-02	.368641E+04	.350588E+03	-.362566E+03
6	6	-.928567E-03	.279984E-03	0.	.296658E+05	-.230317E+03	0.
7	4	-.279984E-03	.928567E-03	-.414563E-17	-.285123E+04	.672351E+03	-.124369E-11
7	5	-.192099E-03	.192099E-03	-.656010E-03	.218369E+04	.207676E+03	-.196803E+03
7	6	.520104E-03	-.135907E-03	0.	.166281E+05	-.723181E+02	0.
7	2	-.135907E-03	.520104E-03	.225028E-17	-.136498E+04	.382331E+03	.675084E-12

(b) Plate forces and stresses for ECC-0.09 case

Figure 23.- Concluded.

74

of the independent variables. The entries in the last row are the values of AC. The variables B(I), T(J), and THET(J) are identified by the numbers in the column denoted VARIABLE NUMBER.

It is important to note here that linking equations are not required in a problem involving only analysis—no sizing. For analysis applications, input values of B, T, and THET are adequate. Linking equations are useful primarily for sizing applications or for parameter studies (a series of single analyses) where dimensions are being changed. Assume, for example, that an engineer wishes to carry out a parameter study with the J-stiffened panel described in this section and that the parameter study involves varying T(1). The offset T(8) of plate elements 4, 5, 12, and 13 varies with T(1). The engineer can either specify the thickness T(1) and the offset T(8) for each case in the parameter study or he can specify T(1) for each case and rely on a linking equation that automatically calculates the corresponding value of T(8).

The plate forces in each plate element and the strains and stresses in each layer are shown in figure 23(b). The columns denoted PLATE NO give the VIPASA identification for the plate elements. Plate numbers 4, 5, and 6 make up the web of the stiffener. The effect of the bending moment can be seen by the variation of the load in these plate elements. The strains and stresses in each layer are parallel and transverse to the fiber direction in that layer.

Example 4, I-Stiffened Panel

The fourth example is an I-stiffened panel loaded by a combination of shear and longitudinal compression as shown i.. figure 24.

The modeling for the I-stiffened panel is similar to that of the J-stiffened panel discussed previously. In the I-stiffened panel example, a fifth linking equation causes the stiffener spacing to be equal to 5.0 during panel sizing.

This example illustrates the adjusted analysis for shear buckling in PASCO. The adjusted shear analysis (rather than the standard VIPASA analysis) is selected by setting the input parameter SHEAR equal to a number greater than zero. The parameter SHEAR is set equal to 1.0 in the input shown in figure 25. If the panel were composed of stiffeners with closed sections, such as hats, a value of shear less than 1.0 is recommended. See definition of SHEAR in section entitled Input and in reference 1.

The program output (fig. 26) selected for this example focuses on the adjusted analysis for buckling when the loading involves shear. The output consists, primarily, of three VIPASA solutions: two solutions obtained with smeared orthotropic stiffnesses and one standard VIPASA solution obtained with discrete stiffeners. These three solutions are used in the adjusted shear analysis and are discussed in the section entitled Adjusted Analysis for Shear Buckling in reference 1. All of this

76

(a) Repeating element

(b) Entire panel, with nodes

$N_{xy} = 600$

$N_x = 6000$
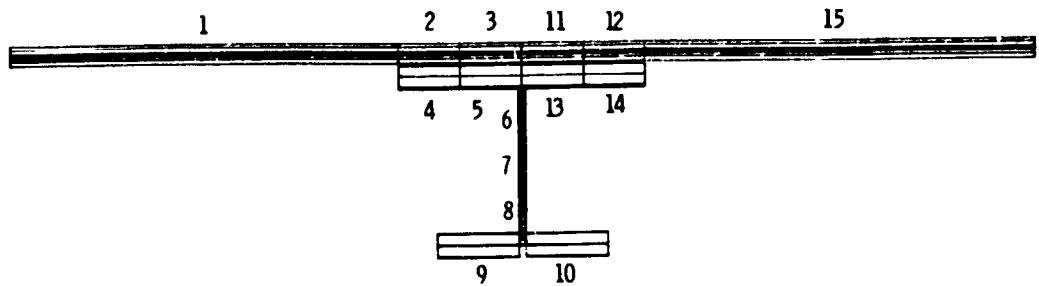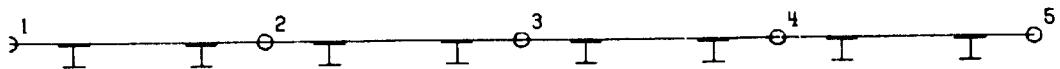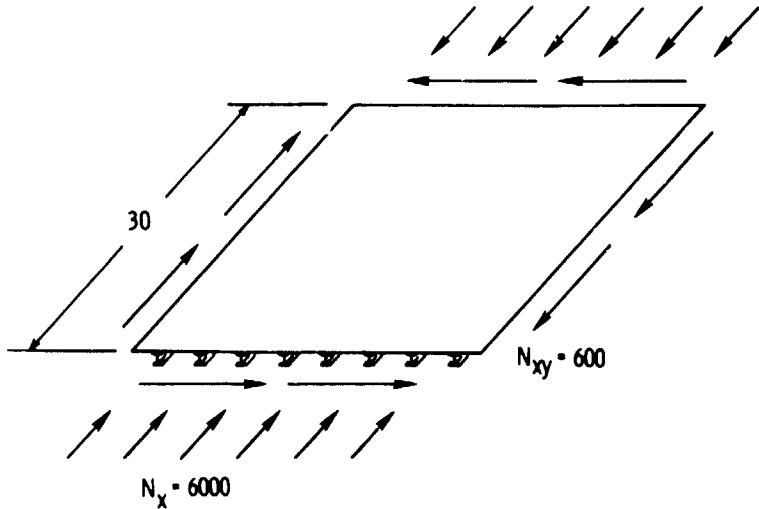
(c) Panel with design loading

Figure 24.- Example 4, I-stiffened panel with eight
stiffeners designed for longitudinal compression and shear.

```
I STIFFENED PANEL
$CONDAT
$
$PANEL
IP=2.
LINK=1.
MAXJJJ=4.
HCARD=  6.-16.-4.0.-8.0.-8.
        6.-17.-5.0.-8.0.-8.
        5.18.2.-16.3.-17.
        6.-19.-13.0.-8.0.-8.
        6.-20.-14.0.-8.0.-8.
        5.21.11.-19.12.-20.
        4.22.8.7.6.
        4.-23.22.90.0.
        6.-24.-23.0.0.0.-9.
        2.121.9.
        2.122.10.
        4.-123.122.180.0.
        6.-124.-121.10.0.0.0.
        6.-125.-123.-10.0.0.0.
        4.126.24.-124.-125.
        6.16.1.18.21.-126.15.
IBC=2.
NLAM(1)=  4.10.
MIMLAM=30.
NFIG(1)=4.
EL=30..
NOBAY=4.   NOSUB=2.
THFT(1)=45..0..90..45..0..90..0..
KWALL(*,1)=1.-1.2.3.
KWALL(*,2)=4.-4.5.6.
KWALL(1.3)=4.-4.7.6.
IWALL=1.1.1.2.2.3.3.3.2.2.1.1.2.2.1.
AT(1.1)=2..1..1..2..1..1..0..-1..
AT(1.2)=2..1..1..4..2..2..0..0..-1..
AB(1.3)=0..1..0.. -1..
AT(1.4)=0..0..0..2..0..1..1..0..0..-1..
AB(1.5)=1..1..1..
AC(5)=2.5.
MAT(1.1)=1.2.1.1.2.1.2.
NX(1)=(000..  NXY(1)=600..
SHFAR=1..
T= -.02. -.006. -.002. -.004. -.05. -.002. -.008.   3*1.E30.
B=1.E30. -.3. .3. 1.E30. .3. -.25. .25. .25. -.4. .4. .3. .3. .3. .3. 1.9.
IORTH=0.
$
$MATER
E1(1)=11.E6. E2(1)=.8E6. E12(1)=.3E6. ANU1(1)=.34. RHO(1)=.05.
ALFA1(1)=.1E-98. ALFA2(1)=1.E-5.
ALLOW(1.1)=2..003.-.017..03-.005..01.
E1(2)=32.E6. E2(2)=2.7E6. E12(2)=.93E6. ANU1(2)=.21. RHO(2)=.072.
ALFA1(2)=.24E-5.  ALFA2(2)=1.15E-5.
ALLOW(1.2)=2..009.-.005..007.-.003..010.
$
```

Figure 25.- Input for example 4, I-stiffened panel with eight stiffeners, designed for longitudinal compression and shear.

ORTHOTROPIC PLATE BUCKLING LOAD WITH X STIFFNESS IN VIPASA X DIRECTION

ITER	NEIG	LAMBDA	FACTOR	LIMITS ON FACTOR		NX	NY	NXY
18	1	.30000000E+02	.37619954E+00	.37619830E+00	.37619995E+00	.225719E+04	0.	.225719E+03

SUMMARY OF CRITICAL WAVELENGTHS AND EIGENVALUES

M	LAMDA	FACTOR
1	30.0000	.3762

$F_{s,0}$

(a) Smeared orthotropic solution, $F_{s,0}$

ORTHOTROPIC PLATE BUCKLING LOAD WITH SIMPLE SUPPORT ALONG THE ENDS AND THE X STIFFNESS IN VIPASA Y DIRECTION

ITER	NEIG	LAMBDA	FACTOR	LIMITS ON FACTOR		NX	NY	NXY
18	1	.40000000E+02	.51633669E+00	.51633606E+00	.51633690E+00	0.	0.	.309802E+03

SUMMARY OF CRITICAL WAVELENGTHS AND EIGENVALUES

M	LAMDA	FACTOR
1	40.0000	.5163

$F_{s,90}$

(b) Smeared orthotropic solution, $F_{s,90}$

Figure 26.- Sample output for example 4, I-stiffened panel with eight stiffeners designed for longitudinal compression and shear.

79

EIGEN... ES CALCULATED TO AN ACCURACY OF CONV) ARE DETERMINED AS FOLLOWS TO IDENTIFY CRITICAL WAVELENGTHS

Standard VIPASA, $F_{d,0}$

ITER	MEIG	LAMBDA	FACTOR	LIMITS ON FACTOR		NX	NY	NXY
12	1	.30000080GE-02	.33160264TE-00	.3316682l0E-00	.33163615OE-00	.199GE-5E-04	0.	.199082E-03
7	1	.75000000GE-01	.74607834lE-00	.74681274GE-00	.74726515E-00	.44618TE-04	0.	.44618TE-03
10	1	.40000000GE-01	.731629816E-00	.73121433lE-00	.731862143E-00	.43897TE-04	0.	.43897TE-03
6	1	.50000000GE-01	.72524202GE-00	.724788997E-00	.72535615GE-00	.43514SE-04	0.	.43514SE-03
21	1	.30000000GE-01	.772095044E-00	.771191113E-00	.72841236E-00	.46325TE-04	0.	.46325TE-03

SUMMARY OF CRITICAL WAVELENGTHS AND EIGENVALUES

M	LAMBDA	FACTOR
1	30.0000	.3318
2	7.5000	.7478
3	5.0000	.7752
4	3.0000	.7721

BECAUSE A NONZERO VALUE OF SHEAR WAS INPUT, THE $M = 1$ BUCKLING LOAD IS CALCULATED USING THE ADJUSTED ANALYSIS WHICH IS BASED ON THE ORTHOTROPIC PLATE THEORY RESULTS OF THE TWO PREVIOUS PROBLEMS.

FACTOR FOR $M = 1$ BUCKLING LOAD

VIPASA F(0.0)	LONG ORTHOTROPIC PLATE F(S,0)	WIDE ORTHOTROPIC PLATE F(S,90)	FD(AS), NASA TM 80181 F(0,90)
.331802856-00	.371C1985-E-08	.516336076-00	.455A722RE-00

PASCO CHOOSES THE SMALLER OF F(S,90) AND F(0,90) FOR CALCULATING THE $M = 1$ BUCKLING LOAD UNLESS IT IS LESS THAN VIPASA, F(0.0). IN WHICH CASE VIPASA, F(0,0) IS USED.

EIGENVALUES FOR CRITICAL WAVELENGTHS ARE NOW CALCULATED TO AN ACCURACY CONV?

ITER	MEIG	LAMBDA	FACTOR	LIMITS ON FACTOR		NX	NY	NXY
15	4	.30000000E-02	.43162906E-00	.4316296OF-00	.43163229CE-00	.3552R5E-04	0.	.3552R5E-03
21	2	.30000000E-02	.37702103E-00	.77072000F-00	.37702290E-00	.3185296-64	0.	.3185296-03
44	2	.30000000E-07	.343644A4NF-00	.343644764F-00	.3434647NAF-00	.24302AF-04	0.	.243026F
10	1	.30000000E-07	.331807437E-00	.331R02917F-00	.331R07910F-00	.27324?E-04	0.	.273247E-03
7	1	.75000000E-01	.746R0154TF-00	.746R1316F-00	.746RR2174T-00	.44R1ROF-04	0.	.44R1ROF-03
7	1	.50000000E-01	.725242A2TF-00	.725246183F-80	.7252ARPN-00	.43514OF-04	0.	.43514OF-03
21	1	.30000000E-01	.772193556F-00	.771031236-06	.772103779F-00	.46331GF-04	0.	.46331GF-03

(c) Standard VIPASA solution, $F_{d,0}$, and adjusted solution

Figure 26.- Continued.

NEGATIVE WALL NUMBERS INDICATE ANISOTROPIC STIFFNESS TERMS NEGLECTED
PLATE FORCES ARE FOR UNIFORM AXIAL STRAIN

PLATE	WALL					
NO.	NC.	BREADTH	NTV	NLV	NSV	B*NLV
1	-1	1.90000	0.00	1612.01	600.00	3062.82
2	-1	.30000	0.00	1612.01	378.99	483.60
3	-2	.30000	0.00	10313.29	221.01	3093.99
4	-3	.25000	0.00	1751.16	0.00	437.79
5	-2	.40000	0.00	10313.29	0.00	4125.31

(d) Stress resultants N_y, N_x, and N_{xy} in each plate element

Figure 26.-- Concluded.

output is for the initial design. Comments added to the output use the same notation as reference 1.

Equation (45), reference 1, is the basic equation used in the adjusted shear analysis. That equation is

$$F_{d,90} = \frac{F_{d,0}}{F_{s,0}} \; F_{s,90} \qquad (10)$$

for this example, the quantities that appear in equation (1) are given by

$$F_{d,90} = \frac{.331802647}{.376198540} \; .516336069$$

$$= .4554022842$$

Since $F_{d,90}$ is smaller than $F_{s,90}$, $F_{d,90}$ is used for the adjusted analysis.

A multiplier defined to be the smaller of

$$\frac{F_{d,90}}{F_{d,0}} \; \text{and} \; \frac{F_{s,90}}{F_{d,0}}$$

is then calculated. In this case, the multiplier is equal to 1.37250947. The adjusted solution is then given by

$$\text{Multiplier} \cdot \text{FACTOR} \cdot [\text{input loading}]$$

82

The multiplier is applied to all eigenvalues for the M=1 mode. For this example, the adjusted solution for the lowest M=1 eigenvalue is

$$1.37250947 \cdot .331802832 \cdot \begin{bmatrix} 6000 \ (N_x) \\ 600 \ (N_{xy}) \end{bmatrix}$$

which gives $N_{x_{cr}} = 2732.41517$, $N_{xy_{cr}} = 273.241517$.

The final output data presented for this example are the forces in each plate. The shear forces are labeled NSV. Note that there are no shear forces in the stiffener.

See additional shear buckling studies in reference 8.

Example 5, Hat-Stiffened Panel

The fifth example is the hat-stiffened panel shown in figure 27. The two design loadings are shown in figure 27(c). Load condition 1 consists of pure longitudinal compression. Load condition 2 consists of longitudinal and transverse compression combined with a 200° increase in temperature.

In the input shown in figure 28, ITHERM(2)=1 allows the panel to take on a bow induced by temperature and transverse load. This bow is treated as an initial bow. Note that material 2, which, in this example, is used for the second load case, has lower stiffness values than material 1. Using this approach, the effect of temperature on material properties can be accounted for. The input also illustrates how the rotation of a plate element can be made to depend upon the lengths of various plate elements making up the cross section. Such an approach, which is based on equation (3), must be used if the lengths of these plate elements change during sizing.

The output selected for this example is the constraint table for the first sizing cycle. The constraint table, which is presented in figure 29, contains the values of the constraints G and the derivatives of the constraints with respect to each sizing variable. The first column in the table identifies the constraints. For example, the first four constraints are the first four eigenvalues for the M=1 mode (λ=L) for load condition 1. The second column gives the values of the constraints G. In this case, the first two constraints are violated, and the second

84

(a) Repeating element

(b) Entire panel, with nodes

LOAD CONDITION 1

$N_x = 4500$

$N_x = 1500$

$N_y = 100$

TEMPERATURE = 200^0,
PANEL ALLOWED TO TAKE
ON INITIAL BOW INDUCED
BY TEMPERATURE AND
TRANSVERSE LOAD

LOAD CONDITION 2
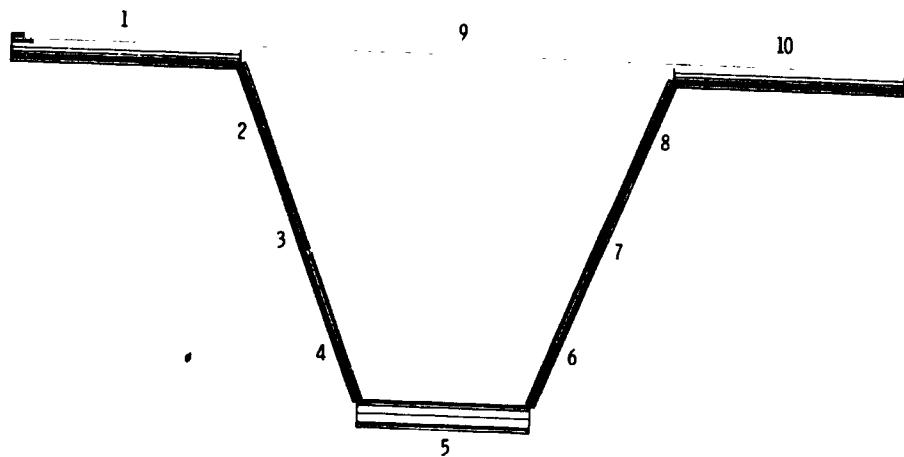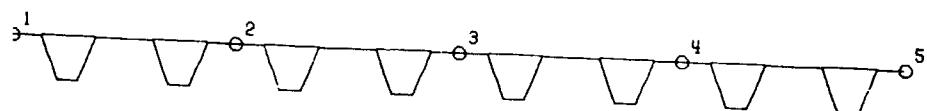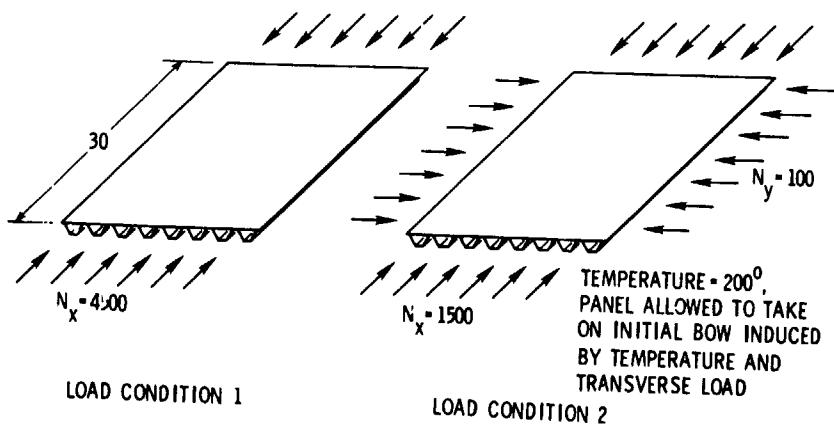
(c) Panel with design loadings

Figure 27.- Example 5, hat-stiffened panel with eight
stiffeners, designed for longitudinal and transverse
compression and temperature.

```
HAT-STIFFFNED PANFL
$CONDAT
$
$PANFL
CLAM=1.0,
LINK=1,
MAXJJJ=4,
IP=2,
BL(5)=.4,    BL(11)=0.001,
B=-.8, -.4, .4  .4, -.6, .4, .4, .4, 1.F30, .4, -.15,
T=-.008, -.016, -.03, 2*1E30,
THET=45,0,0,
AT(1,1)= 2,1,0,-1,
AT(1,2)= 2,0,1,0,-1,
AB(5,3)= 1., 3*0., -1., 0., 6,                    ·
MAT(1,1)= 1, 1, 1,
MAT(1,2)= 2, 2, 2,
ITHERM(2)=1,
KWALL(1,1)=1,-1,-1,1,2,
KWALL(1,2)=1,-1,
KWALL(1,3)=1,-1,3,
IWALL= 1,2,2,2,3,2,2,2,2,1,
EL=30,
NX(1)=4500., 1500.,
NY(2)=100.,
TEM(1,2)=3*200.,
IBC=2,
HCARD=4,12,2,3,4,
       4,13,6,7,8,
       4,-14,12,-11,2,
       4,-15,13,11,2,
       6,-16,-14,0,-4,0,5,
       6,-17,-15,0,5,0,-4,
       6,-18,-9,0,4,0,4,
       4,19,16,5,17,
       5,20,1,18,-19,10,
NOBAY=4,NOSUB=2,
NEIG(1)=4,
MINLAM=30,
$
$MATER
E1(1)= 21F6, E2(1)= 2.39E6, E12(1)= .65F6, ANU1(1)= .314, RHO(1)=.0571,
E1(2)= 19E6, E2(2)= 2.00E6, E12(2)= .60E6, ANU1(2)= .314, RHO(2)=.0571,
ALFA1(1)= -.5F-8,  ALFA2(1)= .218E-4,
ALFA1(2)= -.5F-8,  ALFA2(2)= .218E-4,
ALLOW(1,1)=2, .004, -.004, .004, -.004, .01,
ALLOW(1,2)=2, .004, -.004, .004, -.004, .01,
$
```

Figure 28.- Input for example 5, hat-stiffened panel with eight
stiffeners, designed for longitudinal and transverse
compression and temperature.

86

BUCKLING-LOAD CASE L, AXIAL MODF M, FIGENVALUE J

STRFNGTH-CRITFRIA JJ, LOAD CASF L, PLATE NO. I, LAYFR NO. K, ALLOW(N)

STIFFNESS-LOAD CASE L, M=1-? AXIAL, M=3-4 SHEAR, M=5-6 RENDING, M ODD IS LOWER ROUND, M EVEN IS UPPER ROUND

LLMM.JJ
JJLLIT.KKNN
LLMM

Figure 29.- Sample output for example 5, hat-stiffened panel with eight stiffeners, designed for longitudinal and transverse compression and temperature. Output is the constraint table at the beginning of the first sizing cycle.

87

two constraints are satisfied. The final seven columns are
the derivatives of the constraints with respect to the sizing
variables. The derivatives are given first with respect to $B(I)$,
then $T(J)$, then THET(J). In this case, the order of the
sizing variables is $B(1)$, $B(2)$, $B(5)$, $B(11)$, $T(1)$, $T(2)$,
and $T(3)$ corresponding to the negative signs in the input data
for B and T in figure 28.

Example 6, Z-Stiffened Panel

The sixth example is an aluminum Z-stiffened panel loaded in longitudinal compression as shown in figure 30. Instead of a constraint on buckling, which is used in other examples, in this example a constraint is placed on vibration frequency.

Another feature of this example is the method used to model the attachment of the stiffener to the plate. Whereas in previous examples the attachment simulates a bond, in this example the attachment simulates a seam weld or a line of closely-spaced rivets.

The input is presented in figure 31. Note that since this example involves vibration, RHO must be mass density.

The output selected for this example gives the results of the eigenvalue analysis for the initial design and the CONMIN output for the first sizing cycle. In the eigenvalue output, figure 32, FACTOR is applied to both the input load NX=3000 and the input frequency FREQ=50. For example, in the second eigenvalue for LAMBDA=30, FACTOR is equal to 1.84758441. The value of NX for this eigenvalue is $1.84758441 \cdot 3000 = 5542.75$, which is the first entry in the column labeled NX. The vibration frequency for this eigenvalue is $1.84758441 \cdot 50 = 92.38$, which is not printed out.

In the CONMIN output, figure 33, the default value 1 is used for the print indicator IPRINT. With IPRINT=1, only the initial function information and the final results are printed

(a) Repeating element

(b) Entire panel, with nodes

$N_x = 3000$

● STIFFENERS ATTACHED
ALONG LINE
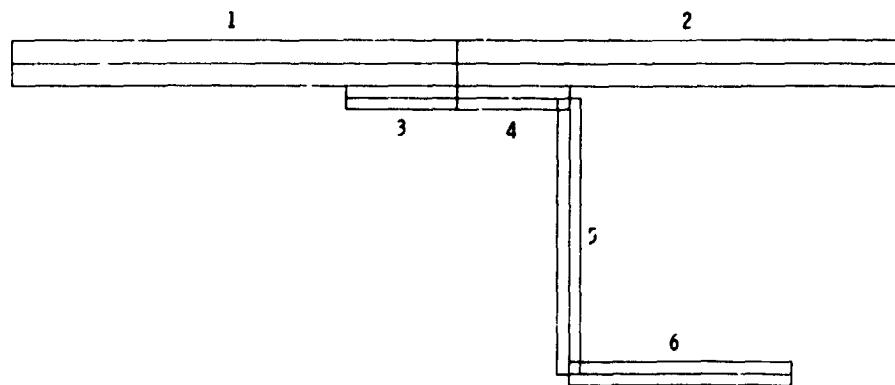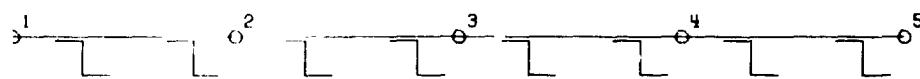
● FREQUENCY CONSTRAINT
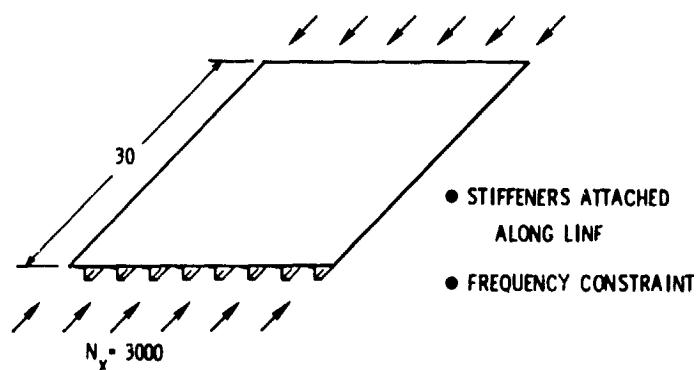
(c) Panel with design loading

Figure 30.- Example 6, Z-stiffened panel with eight
stiffeners, designed for longitudinal compression and
a minimum frequency constraint.

90

```
Z-STIFFENED PANEL
$CONDAT
$
$PANEL
 SMOVE=.1,
 SFACTP=.9,
 LINK=1,
 NFIG=2,
 NLAM=6,8,10,
 MAXJJJ=8,
 IP=2,
 HCARD=6,-7,-3,0,-3,0,-3,
        2,121,7,
        4,-8,4,180,0,
        4,-9,5,90,0,
        4,-10,6,180,0,
        4,122,10,9,8,
        6,-123,-122,0,-3,0,-3,
        5,11,1,2,-121,-123,
 IBC=1,
 FL=30,
 NOBAY=4,     NOSUB=2,
 MINLAM=30,
 THET(1)=0.,0.,
 KWALL(1,1)=1,
 KWALL(1,2)=2,
 IWALL=1,1,2,2,2,2,
 AT(1,1)=1.,1.,-1.,
 AB(1,2)=0.,0.,2.,0.,0.,-1.,
 MAT(1,1)=1,1,
 NX=3000.,
 T=-.1,-.05,1.E30,
 B=-2.,2.,-.5,.5,-1.2,1.E30,
 BL(3)=.5,
 FREQ=50.,
 $
$MATER
 E1=10.5E6,
 E2=10.5E6,
 E12=3.98E6,
 ANU1=.32,
 RHO=.00026138,
 ALFA1=13.E-6,     ALFA2=13.E-6,
 ALLOW=2.,006,-.006,.006,-.006,.01,
 $
```

Figure 31.- Input for example 6, Z-stiffened panel with eight
  stiffeners, designed for longitudinal compression and a
  minimum vibration frequency.

EIGENVALUES FOR CRITICAL WAVELENGTHS ARE ALSO CALCULATED TO AN ACCURACY CONV?

ITER	METH	LAMBDA	FACTOR	LIMITS ON FACTOR		NX	NY	NXY
18	2	.30000000E+02	.18475844IE+01	.18475825IE+01	.16475864E+01	.554275E+04	0.	0.
21	1	.30000000E+02	.16901018TE+01	.16901060E+01	.16901022TE+01	.449458E+04	0.	0.
20	1	.50000000E+01	.74590735IE+01	.74590637E+01	.74590747IE+01	.723772E+05	0.	0.
10	1	.37500000E+01	.72385621E+01	.72385029E+01	.72385824E+01	.217046E+05	0.	0.
10	1	.30800000E+01	.77028635E+01	.77028563E+01	.77028666TE+01	.231686E+05	0.	0.

Figure 32.- Sample output for example 6, Z-stiffened panel. Output gives results of eigenvalue analysis.

```
* * * * * * * * c * * * * * * * * * * * * * * * * * * * *
*                                                        *
*                     C O N M I N                        *
*                                                        *
*              FORTRAN PROGRAM FOR                       *
*                                                        *
*         CONSTRAINED FUNCTION MINIMIZATION              *
*                                                        *
* * * * * * * * * * * * * * * * * * * * * * * * * * * * *
```

INITIAL FUNCTION INFORMATION

OBJ =   .243955E-05

DECISION VARIABLES (X-VECTOR)
   1)   .20000E+01   .50000E+00   .12000E+01   .10000E+00   .50000E-01

CONSTRAINT VALUES (G-VECTOR)
   1)   -.49819E+00   -.84758E+00

FINAL OPTIMIZATION INFORMATION

 OBJ =   .211718E-05

DECISION VARIABLES (X-VECTOR)
   1)   .22000E+01   .50000E+00   .10800E+01   .90000E-01   .45000E-01

CONSTRAINT VALUES (G-VECTOR)
   1)   -.46069E-01   -.24795E+00

THERE ARE   0 ACTIVE CONSTRAINTS

THERE ARE   0 VIOLATED CONSTRAINTS

THERE ARE   5 ACTIVE SIDE CONSTRAINTS
DECISION VARIABLES AT LOWER OR UPPER BOUNDS (MINUS INDICATES LOWER BOUND)
     1   -2   -3   -4   -5

TERMINATION CRITERION
     ABS(OBJ(I)-OBJ(I-1))   LESS THAN DABFUN FOR   8 ITERATIONS

NUMBER OF ITERATIONS =   8

OBJECTIVE FUNCTION WAS EVALUATED   12 TIMES

CONSTRAINT FUNCTIONS WERE EVALUATED   12 TIMES

GRADIENT OF OBJECTIVE WAS CALCULATED   5 TIMES

GRADIENTS OF CONSTRAINTS WERE CALCULATED   5 TIMES

Figure 33.- Sample output for example 6, Z-stiffened panel.
          Output is from CONMIN program.

93

in CONMIN. In the final optimization information, the quantities labeled CONSTRAINT VALUES (G-VECTOR) are approximate values of the constraints calculated using the approximate analysis approach discussed in reference 1.

Example 7, Truss-Core Sandwich Panel

The seventh example is a truss-core sandwich panel loaded by a combination of longitudinal compression and lateral pressure. The panel and loading are shown in figure 34. The example is relatively complex and, in some respects, stretches to the utmost the capability of the program in order to illustrate its generality and limitations.

The input is shown in figure 35. Because of limitations associated with HCARD input discussed earlier, ICARD input is required to model the panel. Even though HCARD input is not adequate to model the repeating element from the point of view of correct structural response, HCARD input can be used to model the geometry of the repeating element. Because the lateral pressure loading causes a bending moment, the geometry of the repeating element must be defined by HCARD input in order that D_{11} and the longitudinal load distribution N_{x_i} be correct.

The single linking equation states that $B(7) = .5 B(1)$. This value of $B(7)$ is used in the rotations specified by the first four HCARD sequences. Plate elements 7 to 10 created by these first four HCARD sequences are used for ICARD input. The other HCARD sequences illustrate one of several ways to construct the geometry of the repeating element.

The output selected for this example is shown in figure 36. The upper portion of the figure is the HCARD and ICARD input actually used by VIPASA. Note the large amount of ICARD data

C-2

(a) Repeating element

(b) Entire panel, with nodes

$P = 15$

30

$N_x = 18000$
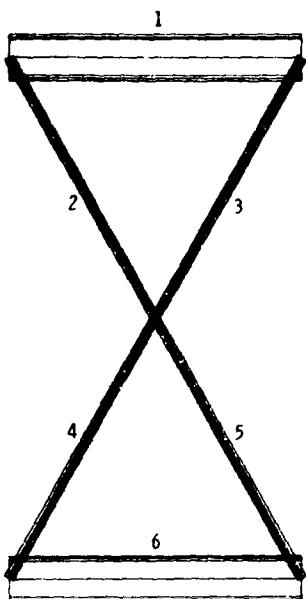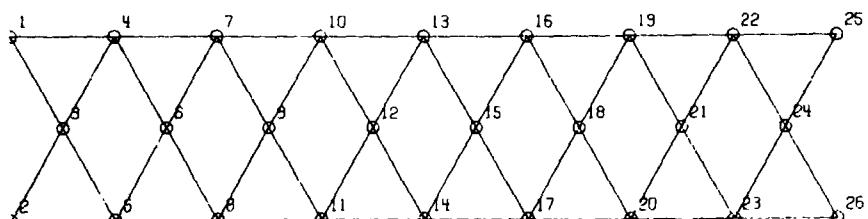
(c) Panel with design loading

Figure 34.- Example 7, truss-core sandwich panel with eight repeating elements, designed for longitudinal compression and lateral pressure.

96

```
****** TRUSS CORE PANEL*****
$CONDAT
$
$PANEL
MAXJJJ=8,
LINK=1,
IP=2,
MINLAM=30.,
PRESS=15.,
NX=18000.,
THFT=45,45,0,
R=-1., -1., 1., 1., 1., 1., 1.E30,
T=-.008, -.008, -.06,
TL=.0055, .0055, .0055,
KWALL(1,1)=1.-1.3,
KWALL(1,2)=2.-2,
IWALL=1,2,2,2,2,1,
EL=30,
HCARD=4,-7,2,-7,2,
      4,-8,3,7,2,
      4,-9,4,7,2,
      4,-10,5,-7,2,
      4,-11,10,180,0,
      3,12,6,11,
      3,121,9,-12,
      4,13,7,8,-121,
      3,122,1,-13,
IRAY=3,   NOBAY=8,    ICREP=8,
ICARD=7,1,3,7,4,1,-909,900,
      7,2,3,9,5,6,-909,900,
      5,3,4,8,5,10,
      3,4,-909,900,
      3,5,-909,900,
AR(1,1)=.5, 5*0., -1.,
$
$MATER
F1(1)=19.E6,  F2(1)=1.89E6,  E12(1)=.93E6,   ANU1(1)=.31,  RHO(1)=.0571,
ALFA1=-.5E-8,ALFA2=.218E-4,
ALLOW=2,.004,-.004,.004,-.004,.01,
$
```

Figure 35.- Input for example 7, truss-core sandwich panel
designed for longitudinal compression adn lateral pressure.

97

98

SUB-STRUCTURES ARE DEFINED AS FOLLOWS

```
4   -7    2   -7    2
4   -8    2    7    2
4   -9    3    7    2
4  -10    3   -7    2
4  -11   10  180    0
3   12    4   11
3  121    9  -12
4   13    7    8 -121
3  122    1  -13
```

THE FINAL STRUCTURE IS DEFINED AS FOLLOWS

```
7    1    3    7    4    5          1-909 900
7    2    3    9    8    5          4-909 900
5    3    4    6    7    5    10
5    4    5    6    7    9     1
5    5    5    7    8   10     4
5    6    6    7    9    8    10
5    7    9    9    7    8     1
5    8    9   10    8   11    10
5    9   10   12    7    9     1
5   10   11   12    8   11     4
5   11   12   13    7    9    10
5   12   13   15    8   11     1
5   13   14   16    7    9    10
5   14   15   17    8   11     4
5   15   16   18    7    9    10
5   16   17   19    8   11     1
5   17   18   20    7    9    10
5   18   19   21    8   11     4
5   19   20   22    7    9    10
5   20   21   23    8   11     1
5   21   22   24    7    9    10
5   22   23   25    8    9     4
5   23   24   26    7    9    10
5   24   25   26    8   11     1
3   25 -909 900
3   26 -909 900
```

OBJ = MASS/(AREA*LENGTH) = .822240E-03
TOTAL PANEL MASS = .592013E+01
PANEL WIDTH = .800000E+01

SMEARED PANEL STIFFNESSES ASSIGNED TO PLATE 16

A11	A22	A33	D11	D22	D33	ZBAR	AW
.520757E+07	-.921304E+04	-.104142E+03	.370914E+07	-.193241E-29	.144237E+06	.866025E+00	.266675E-01

Figure 36.— Sample output for example 7, truss-core sandwich panel. Output shows HCARD and ICARD modeling used by VIPASA and smeared orthotropic stiffnesses of panel.

here, figure 36, compared with the small amount in the input, figure 35.

At the bottom of figure 36 are six smeared orthotropic stiffnesses for the panel and quantities denoted ZBAR and AM. The stiffnesses are defined in reference 1. The stiffnesses are calculated from the repeating element constructed from HCARD input, and because of limitations in that model, are not all correct. In particular, the values given for the stiffnesses A22, A33, and D22, which are calculated by VIPASA, are meaningless since the repeating element is defined as being singly connected. With the present limitations in VIPASA, no modeling approach would provide correct values for A22, A33, and D22. The values given for A11 and D11 are correct. The value given for D33 is too small. The value is approximately the twisting stiffness of the upper triangle of the repeating element. Since several pertinent stiffnesses are incorrect, the adjusted shear analysis (example 4) should not be used for this configuration. It is important to recognize that these smeared stiffnesses are not used in VIPASA, and, therefore, errors in the stiffnesses do not cause errors in the VIPASA buckling analysis.

The quantity ZBAR is the distance from the reference surface to the centroid of the cross section. The reference surface is the horizontal plane that passes through the first node of the repeating element. The quantity AM is the mass per unit area of the panel. Both values are correct.

Example 8, Stringer-Stiffened Cylinder

The eighth example is a composite cylinder stiffened by rectangular, integral stringers ("blades") and loaded by longitudinal compression. The cylinder and loading are shown in figure 37.

The input is shown in figure 38. HCARD input is used to create eight sections of the stiffened cylinder. ICARD input is used (and is required) to connect these eight sections to form the cylinder. The last three HCARD sequences create the repeating element. The repeating element is used to calculate the stiffnesses and the value of the uniform strain ϵ_x that is used for all plate elements - including the plate elements in the cylinder. Note that in order to calculate the stiffnesses and ϵ_x, the repeating element is flat. Note also that the repeating element is not used in the cylinder.

The input also illustrates that substructure numbers can be reused, and that the B(I) used in rotations are the original values of B(I) and are not associated in any way with subsequent substructures having the same value of I. In this example, B(7) to B(12) are cosines of the desired angles. As can be seen in the input, the skin on the right half of the cylinder is constructed of wall 1, and the skin on the left half of the cylinder is constructed of wall 3. Wall 3 is identical with wall 1 except that the angles are reversed. The cylinder is modeled in this way so that after rotation of plate elements used on the two halfs of the cylinder, the ply angles in each

100
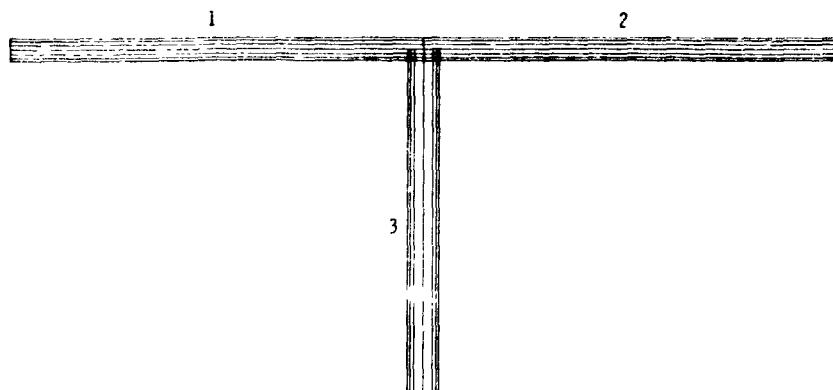
(a) Repeating element
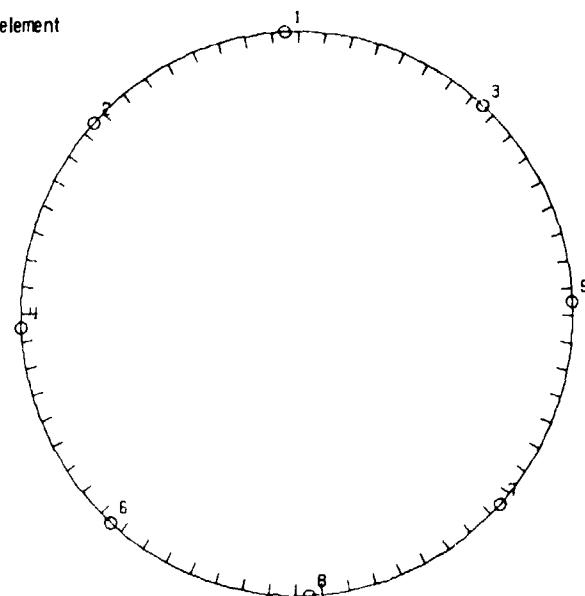
(b) Entire panel (cylinder), with nodes
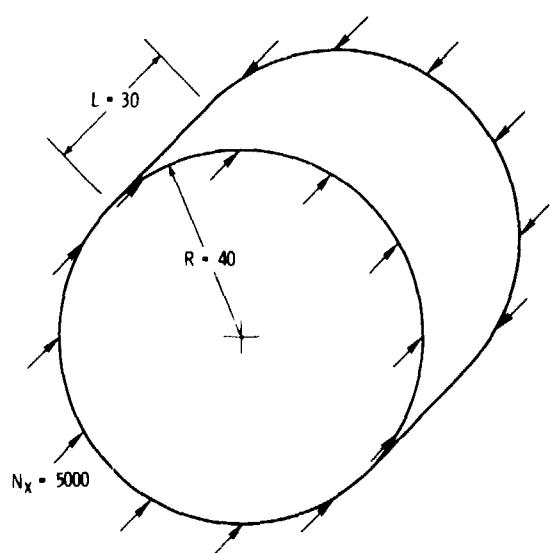
(c) Cylinder with design loading

Figure 37.- Example 8, composite stringer-stiffened cylinder
with radius equal to 40, length equal to 30, designed for
longitudinal compression.

101

```
COMPOSITE, STRINGER-STIFFENED CYLINDER
$CONDAT ALPHAX=.04,
$
$PANEL
LINK=1,
SMOVE=.1,
MAXJJJ=3,
IORTH=1,
IP=2,
NX=5000,
NLAM=7,
MTNLAM=30,
NEIG=30*4,
NOSUB=1,  NOBAY=64,
B(1)=2*1.96349E+09,-1.59,.9817477042,1.59,.9817477042,.9999247018,
     .9993223846,.9951847267,.9807852804,.9238795325,.7071067812,1.,
T=-.0147,-.0232,-.0161,-.0435,
THET=45,0,45,
KWALL(1,1)=1,-1,2,
KWALL(1,2)=3,-3,4,
KWALL(1,3)=-1,1,2,
IWALL=1,1,2,1,2,3,
EL=30.,
HCARD=4,-7,4,8,13,           4,-8,4,7,13,
      4,-9,4,-7,13,          4,-10,4,-8,13,
      4,-11,5,90,0,          2,121,11,
      6,12,7,8,9,-121,10,    4,-13,12,-9,13,
      3,14,12,13,            4,-15,14,-10,13,
      3,16,14,15,            4,-17,16,-11,13,
      3,18,16,17,            4,-19,18,-12,13,
      4,-20,19,-12,13,       4,-21,20,-12,13,
      4,-7,6,8,-13,          4,-8,6,7,-13,
      4,-9,6,-7,-13,         4,-10,6,-8,-13,
      6,11,7,8,9,-121,10,    4,-12,11,9,13,
      4,-13,12,9,13,         3,14,12,13,
      4,-15,14,10,13,        3,16,14,15,
      4,-17,16,11,13,        3,22,16,17,
      4,-23,22,12,13,        4,-24,23,12,13,
      4,-25,24,12,13,        4,-10,3,90,0,
      2,121,10,              4,15,1,2,-121,
ICARD=5,1,3,18,2,22,         3,2,4,23,
      3,3,5,19,              3,4,6,24,
      3,5,7,20,              3,6,8,25,
      3,7,8,21,
$
$MATER E1=19.0E6,E2=1.89E6,E12=.93E6,ANU1=.31,RHO=.0571,
ALFA1=-.5E-8,ALFA2=.218E-4,
ALLOW=2,.005,-.005,.005,-.005,.01,
$
```

Figure 38.- Input for example 8, stringer-stiffened
crylinder designed for longitudinal compression.

layer are continuous. Differences in the walls ($\pm\theta$ vs. $\mp\theta$) would not be detected if anisotropic bending stiffnesses were ignored, but with IORTH=1, as it is in this example, tne differences between walls can be detected.

The output selected for this example consists of data describing the plate elements, figure 39(a), and HCARD and ICARD input data actually used by VIPASA, figure 39(b). The data describing the plate elements gives the VIPASA plate number assigned to the plate element widths, and shows at a glance that B(3) is the only plate element width that is a sizing variable. The HCARD input shown in figure 39(b) uses the VIPASA plate numbers given in figure 39(a). The ICARD input in figure 39(b) is identical to the ICARD input presented in figure 38.

IN THE FOLLOWING--
ASTERISKS PRINTED FOR   AL,   TL,   AND   THETL   INDICATE THE ASSOCIATED DIMENSION IS NOT A DESIGN VARIABLE

THE INTEGER SUBSCRIPTS HAVE THE FOLLOWING MEANING:

I   REFERS TO PLATE
IM   REFERS TO WALL NUMBER
J   REFERS TO GENERAL LAYER J
K   REFERS TO THE KTH LAYER IN A GIVEN WALL
L   REFERS TO A LOAD CASE L
M   REFERS TO AXIAL HALF-WAVE NUMBER
MA   REFERS TO MATERIAL MA
N   REFERS TO EQUATION NUMBER N LINKING VARIABLES

PLATE ELEMENT WIDTHS

VARIABLE NO.	P(I)	LOWER BOUND BL(I)	UPPER BOUND BU(I)	WALL NO.	VIPASA PLATE NO.	FNXY(I)	FNY(I)
1	1.963495	**********	.100000E+11	1	1	0.000000	0.000000
2	1.963495	**********	.100000E+11	1	2	0.000000	0.000000
3	1.590000	0.000	.100000E+11	2	3	0.000000	0.000000
4	.981748	**********	.100000E+11	1	2	0.000000	0.000000
5	1.590000	**********	.100000E+11	2	4	0.000000	0.000000
6	.981748	**********	.100000E+11	3	0	0.000000	0.000000
7	.999925	**********	.100000E+11	0	0	0.000000	0.000000
8	.999322	**********	.100000E+11	0	0	0.000000	0.000000
9	.995185	**********	.100000E+11	0	0	0.000000	0.000000
10	.980785	**********	.100000E+11	0	0	0.000000	0.000000
11	.923880	**********	.100000E+11	0	0	0.000000	0.000000
12	.707107	**********	.100000E+11	0	0	0.000000	0.000000
13	1.000000	**********	.100000E+11	0	0	0.000000	0.000000

(a) Data describing plate elements, including VIPASA plate numbers

Figure 39.- Sample output for example 8, stringer-stiffened cylinder.

```
SUB-STRUCTURES ARE DEFINED AS FOLLOWS
  4   -7    3    8   13
  4   -8    3    7   13
  4   -9    3   -7   13
  4  -10    3   -8   13
  4  -11    2   90    0
  2  121   11
  6   12    7    8    9-121   10
  4  -13   12   -9   13
  3   14   12   13
  4  -15   14  -10   13
  3   16   14   15
  4  -17   16  -11   13
  3   18   16   17
  4  -19   18  -12   13
  4  -20   19  -12   13
  4  -21   20  -12   13
  4   -7    4    8  -13
  4   -8    4    7  -13
  4   -9    4   -7  -13
  4  -10    4   -8  -13
  6   11    7    8    9-121   10
  4  -12   11    9   13
  4  -13   12    9   13
  3   14   12   13
  4  -15   14   10   13
  3   16   14   15
  4  -17   16   11   13
  3   22   16   17
  4  -23   22   12   13
  4  -24   23   12   13
  4  -25   24   12   13
  4  -10    2   90    0
  2  121   10
  4   15    1    1-121
THE FINAL STRUCTURE IS DEFINED AS FOLLOWS
  5    1    3   18    2   22
  3    2    4   23
  3    3    5   19
  3    4    6   24
  3    5    7   20
  3    6    8   25
  3    7    8   21
```

(b) HCARD and ICARD data used by VIFASA

Figure 39.- Concluded.

REFERENCES

1. Stroud, W. Jefferson; and Anderson, Melvin S.: PASCO: Structural Panel Analysis and Sizing Code, Capability and Analytical Foundations. NASA TM-80181, 1981. (Supersedes NASA TM-80181, 1980)

2. Wittrick, W. H.; and Williams, F. W.: Buckling and Vibration of Anisotropic or Isotropic Plate Assemblies Under Combined Loadings. Int. J. Mech. Sci., Vol. 16, 1974, pp. 209-239.

3. Williams, F. W.; and Anderson, M. E.: Users' Guide to VIPASA (Vibration and Instability of Plate Assemblies Including Shear and Anisotropy). Department of Civil Engineering, University of Birmingham (England), January 1973

4. Anderson, Melvin S.; Hennessy, Katherine W.; and Heard, Walter L., Jr.: Addendum to Users' Guide to VIPASA (Vibration and Instability of Plate Assemblies Including Shear and Anisotropy). NASA TM X-73914, 1976.

5. Vanderplaats, Garret N.: CONMIN - A Fortran Program for Constrained Function Minimization. User's Manual. NASA TM X-62,282, 1973.

6. Vanderplaats, G. N.; and Moses, F.: Structural Optimization by Methods of Feasible Directions. National Symposium on Computerized Structural Analysis and Design, Washington, DC, March 1972.

7. Stroud, W. Jefferson; Greene, William H.; and Anderson, Melvin S.: Current Research on Shear Buckling and Thermal Loads With PASCO: Panel Analysis and Sizing Code. NASA TM-83206, 1981.

8. Stroud, W. Jefferson; Greene, William H.; and Anderson, Melvin S.: Buckling Loads for Stiffened Panels Subjected to Combined Longitudinal Compression and Shear Loadings: Results Obtained With PASCO, EAL, and STAGS Computer Programs. NASA TM-83194, 1981.

106